

Cadence Orcad Guide

Tutorial OrCAD and Cadence Allegro PCB Editor | 2022 | Step by Step | For Beginners Starting with OrCAD and Cadence Allegro PCB - Tutorial for Beginners Cadence OrCad Capture 17.4 - Detailed Overview Tutorial Follow The Blue Knights Audio Engineer For A Day 3 Simple Tips To Improve Signals on Your PCB - A Big Difference How To Improve Your PCB Designs (Common Mistakes) - Phil's Lab #18 Making of PCBs at home, DIY using inexpensive materials From Idea to Schematic to PCB - How to do it easily! Printed Circuit Board Design : Beginner. Step by step OrCAD Introduction - DC Circuit Cadence OrCad 17.4 PSPICE - TCL Scripting, Running Simulations Which PCB Design Software is The Best? The Top 4 Are PSPICE ORCAD Tutorial Part II: Op-Amps Cadence OrCad PCB Editor 17.4 (Complete board layout in 30 min) OrCAD How-To - Using the Cadence Help Tutorial Cadence OrCAD Allegro How to install Cadence OrCAD and Allegro 22.1||Latest methode|| 2023 [New Release] Intro to Cadence OrCad 17.4 Efficient Reference Designator Update in OrCAD Capture: Step-by-Step Guide OrCAD How-To Capture Library Management Tutorial Cadence OrCAD Allegro Cadence OrCad PCB Editor 17.4 (Creating Custom Footprints) Doing PCB Layout - Learn OrCAD \u0026 Cadence Allegro Essentials (Lesson 9) OrCAD How-To Shape Void correct Tutorial Cadence OrCAD Allegro Tutorial OrCAD Allegro Finding Nets and Components

OrCAD Downloads | OrCAD - OrCAD | Cadence

OrCAD PSpice Designer - OrCAD | Cadence

Cadence | Computational Software for Intelligent System ...

Starting with OrCAD and Cadence Allegro PCB - Tutorial for Beginners [Installation guide for OrCAD/Allegro 2019](#) [\[\[OrCAD 17.2Tutorial\]](#)

[How to intalling OrCAD 17.2 \[full active\]](#) [Tutorial Cadence HELP OrCAD and Allegro](#) **Complete PCB Design Tutorial [2019] |**

OrCAD/Allegro 17.2 [How to build and simulate a simple circuit in PSpice?](#) | Srikesh Nagoji [OrCAD How-To Set Up CIS Tutorial](#)

[Cadence OrCAD Allegro](#) **Doing PCB Layout - Learn OrCAD \u0026 Cadence Allegro Essentials (Lesson 9) Installation Guide**

of Orcad Cadence v17.4 Electrical Design Software #orcad#cadence#install#guide [OrCad 17.2 Full Installation](#) [Cadence](#)

[OrCAD Tutorial - How to generate PCB \(board\) in OrCAD 17.4](#) [Install Cadence SPB Allegro and OrCAD 2020 Full Working 100% Printed](#)

Circuit Board Design : Beginner. Step by step [Making of PCBs at home, DIY using inexpensive materials](#) [OrCAD PsPICE 16.6](#)

[Tutorial 0: Descarga e instalación de OrCAD](#) [Intro to Cadence 1: Creating a Schematic and Symbol](#) [How to download and install I](#)

[Pspice student version 9.1](#) **How to start with Cadence Allegro - Very simple tutorial Complete [Arduino UNO] Schematic in**

OrCad Capture / Allegro (2019) [OrCAD Tutorial #1 - Come creare uno schema elettrico, OrCAD PCB Designer 17.4](#)

Cadence Allegro Tutorial - How to create SKILL Script and your own Commands [Instalar OrCAD - Installing OrCAD](#) [Pspice-Cadence-SPB](#)

[OrCAD + Crack \[Google Drive+Mega\]](#)[\[\[Install Guide\]](#) [How To Install Orcad 16.6](#) [Cadence OrCAD simple flow from schematic to PCB](#)

[Cadence OrCAD / Allegro Design Entry CIS Demo Tutorial \(Part 1\)](#) [Allegro orCAD 17-2 Installation process](#) | [Download Link OrCAD](#)

[Allegro How-To create complex footprints Tutorial](#) [OrCAD-Cadence Allegro Tutorial](#) [Cadence OrCAD and Allegro PCB Editor - Change](#)

[Text Line Thickness OrCAD Capture Tutorial in Tamil](#) || [Schematic Design](#) || [Cadence](#) || [NewTechnify](#)

Software Downloads - Cadence

Tutorials | OrCAD

Tutorial paso a paso ¿Cómo Instalar OrCAD 17.4 Full ...

PCB Design Software | OrCAD | Cadence

Install Cadence OrCAD 17.4 : EMA Technical Support

Cadence Orcad Guide

Install OrCAD with Student Licensing : EMA Technical Support

Quick Start Guide: Installing Cadence Allegro/OrCAD via ...

How to Use Multiple Batteries in a Circuit: A Beginner's Guide

32 Bit Microcontroller: A Beginner's Guide - Cadence Blog

Optimization Guide for PCB Routing Constraints - Cadence Blog

Your PDN Design Guide - Cadence Design Systems

Cadence Allegro and OrCAD 17.2-2016 Installation Guide for ...

Cadence SPB/OrCAD 16.6 version 16.60.062 by Cadence Design ...

LAYARUG.PDF - \u00ae Layout Orcad Autorouter User\u2019s ...

GRANT LEONIDAS

OrCAD Downloads | OrCAD - OrCAD | Cadence *Starting with*

OrCAD and Cadence Allegro PCB - Tutorial for Beginners

[Installation guide for OrCAD/Allegro 2019](#) [\[\[OrCAD 17.2Tutorial\]](#)

[How to intalling OrCAD 17.2 \[full active\]](#) [Tutorial Cadence HELP](#)

[OrCAD and Allegro](#) **Complete PCB Design Tutorial [2019] |**

OrCAD/Allegro 17.2 [How to build and simulate a simple circuit](#)

[in PSpice?](#) | Srikesh Nagoji [OrCAD How-To Set Up CIS Tutorial](#)

[Cadence OrCAD Allegro](#) **Doing PCB Layout - Learn OrCAD**

\u0026 Cadence Allegro Essentials (Lesson 9) Installation

Guide of Orcad Cadence v17.4 Electrical Design Software

#orcad#cadence#install#guide [OrCad 17.2 Full Installation](#)

[Cadence OrCAD Tutorial - How to generate PCB \(board\) in OrCAD](#)

[17.4](#) [Install Cadence SPB Allegro and OrCAD 2020 Full Working](#)

100% Printed Circuit Board Design : Beginner. Step by

step [Making of PCBs at home, DIY using inexpensive materials](#)

[OrCAD PsPICE 16.6 Tutorial 0: Descarga e instalación de OrCAD](#)

[Intro to Cadence 1: Creating a Schematic and Symbol](#) [How to](#)

[download and install I](#) [Pspice student version 9.1](#) **How to start**

with Cadence Allegro - Very simple tutorial Complete

[Arduino UNO] Schematic in OrCad Capture / Allegro

(2019) [OrCAD Tutorial #1 - Come creare uno schema elettrico,](#)

[OrCAD PCB Designer 17.4](#)

Cadence Allegro Tutorial - How to create SKILL Script and your own Commands [Instalar OrCAD - Installing OrCAD Pspice](#) Cadence SPB OrCAD + Crack [Google Drive + Mega] [Install Guide] [How To Install Orcad 16.6 Cadence OrCAD simple flow from schematic to PCB Cadence OrCAD / Allegro Design Entry CIS Demo Tutorial \(Part 1\) Allegro orCAD 17.2 Installation process | Download Link OrCAD Allegro How To create complex footprints Tutorial OrCAD Cadence Allegro Tutorial Cadence OrCAD and Allegro PCB Editor - Change Text Line Thickness OrCAD Capture Tutorial in Tamil || Schematic Design || Cadence || NewTechnify](#) Cadence Orcad Guide Quick Start Guide: Installing Cadence Allegro/OrCAD via CADpass R20 Prerequisites for Installing Allegro/OrCAD with CMC License Manager System (LMS). Have a unique login name (LINUX or... System Requirements. One of the Cadence-supported operating systems is required. The following is recommended by ... Quick Start Guide: Installing Cadence Allegro/OrCAD via ... The "Cadence License Server Configuration Utility" is used to update your licensing for OrCAD products after you modify your purchase of OrCAD products or services you are licensed to use. If your updated license file is on a dongle, be sure the Cadence licensing dongle is plugged into your computer before you begin. Install Cadence OrCAD 17.4 : EMA Technical Support OrCAD PCB designer, by Cadence, allows you to navigate through these challenges by setting and visualizing constraints in real-time, as well as saving constraints for future projects to avoid starting from scratch. If you're looking to learn more about how Cadence has the solution for you, talk to us and our team of experts. About the Author 32 Bit Microcontroller: A Beginner's Guide - Cadence Blog One such tool from Cadence is the IR Drop Vision integrated analysis tool available in Allegro PCB Designer. This tool lets you see the power integrity status of your PCB design in real-time. It will show you pins that are not receiving a sufficient amount of voltage on a power net so that you can make immediate corrections to your layout. Your PDN Design Guide - Cadence Design Systems The "Cadence License Server Configuration Utility" is used to update your licensing for OrCAD products after you modify either your purchase of OrCAD products or the services you are licensed to use. Open your Windows Start menu and select Cadence > License Server Configuration Utility to launch the utility. Install OrCAD with Student Licensing : EMA Technical Support Before you begin Welcome to OrCAD OrCAD offers a total solution for your core design tasks: schematic- and VHDL-based design entry; FPGA and CPLD design synthesis; digital, analog, and mixed-signal simulation; and printed circuit board layout. What's more, OrCAD's products are a suite of applications built around an engineer's design flow—not just a collection of independently developed ... LAYARUG.PDF - \u00ae Layout Orcad Autorouter User \u2019s ... Cadence Allegro and OrCAD products are integrated directly with Windows; the products support hardware and peripherals supported by Windows. A list of hardware and peripherals officially supported by Windows can be obtained from the Microsoft web page. The products require updating certain Microsoft libraries in the Windows directory. Cadence Allegro and OrCAD 17.2-2016 Installation Guide for ... Describes the design cycle for an electronic design, starting with capturing the electronic circuit in OrCAD Capture, simulating the design with PSpice, through the PCB layout stages in OrCAD Layout / OrCAD PCB Editor, and SPECCTRA, and finishing with the processing of the manufacturing output and maintaining the design through ECO cycles. To enable users to evaluate the power of the OrCAD PCB tools used in the Windows-based PCB design process. Tutorials | OrCAD Customize Your PCB Design Experience with OrCAD new features like design reuse, advanced PCB routing technologies, In-design DFM, Interactive 3D Canvas PCB Design Software |

OrCAD | Cadence Cadence is a leading EDA and Intelligent System Design provider delivering hardware, software, and IP for electronic design. Cadence | Computational Software for Intelligent System ... The Cadence® OrCAD® /Allegro® FREE Physical Viewer is a free download that allows you to view and plot databases from OrCAD PCB Editor, Allegro PCB Editor, Allegro Package Designer, and Allegro PCB SI technology. OrCAD Downloads | OrCAD - OrCAD | Cadence Archivos (Orcad+Crack): <http://raboninco.com/oe1> (1 solo acortador de enlace) Allegro Crack Master (Limpiador): <http://raboninco.com/10hcf> (1 solo acortador de ... Tutorial paso a paso ¿Cómo Instalar OrCAD 17.4 Full ... About the Author Cadence PCB solutions is a complete front to back design tool to enable fast and efficient product creation. Cadence enables users accurately shorten design cycles to hand off to manufacturing through modern, IPC-2581 industry standard. Follow on LinkedIn Visit Website More Content by Cadence PCB Solutions Optimization Guide for PCB Routing Constraints - Cadence Blog OrCAD® PSpice® and Advanced Analysis technology combine industry-leading, native analog, mixed-signal, and analysis engines to deliver a complete circuit simulation and verification solution. Whether you're prototyping simple circuits, designing complex systems, or validating component yield and reliability, OrCAD PSpice technology provides ... OrCAD PSpice Designer - OrCAD | Cadence About the Author Cadence PCB solutions is a complete front to back design tool to enable fast and efficient product creation. Cadence enables users accurately shorten design cycles to hand off to manufacturing through modern, IPC-2581 industry standard. Follow on LinkedIn Visit Website More Content by Cadence PCB Solutions How to Use Multiple Batteries in a Circuit: A Beginner's Guide Cadence SPB/OrCAD 16.6 is a computer program. This page holds details on how to uninstall it from your PC. It was created for Windows by Cadence Design Systems, Inc.. Additional info about Cadence Design Systems, Inc. can be read here. Cadence SPB/OrCAD 16.6 version 16.60.062 by Cadence Design ... InstallScape is a Cadence application which facilitates the downloading and installation of Cadence software in a single process. The selected products can then be saved in a local Archive directory. Go to Downloads to obtain InstallScape, access whitepapers, user manuals, and more. Allegro FREE Physical Viewer Software Downloads - Cadence Trademarks: Trademarks and service marks of Cadence Design Systems, Inc. contained in this document are attributed to Cadence with the appropriate symbol. For queries regarding Cadence's trademarks, contact the corporate legal department at the address shown above or call 800.862.4522. About the Author Cadence PCB solutions is a complete front to back design tool to enable fast and efficient product creation. Cadence enables users accurately shorten design cycles to hand off to manufacturing through modern, IPC-2581 industry standard. Follow on LinkedIn Visit Website More Content by Cadence PCB Solutions OrCAD PSpice Designer - OrCAD | Cadence Cadence is a leading EDA and Intelligent System Design provider delivering hardware, software, and IP for electronic design. **Cadence | Computational Software for Intelligent System ...** OrCAD® PSpice® and Advanced Analysis technology combine industry-leading, native analog, mixed-signal, and analysis engines to deliver a complete circuit simulation and verification solution. Whether you're prototyping simple circuits, designing complex systems, or validating component yield and reliability, OrCAD PSpice technology provides ...

STARTING WITH ORCAD AND CADENCE ALLEGRO PCB - TUTORIAL FOR BEGINNERS INSTALLATION GUIDE FOR ORCAD/ALLEGRO 2019 [[ORCAD 17.2 TUTORIAL] HOW TO INSTALLING ORCAD 17.2 [FULL ACTIVE] TUTORIAL CADENCE HELP ORCAD AND ALLEGRO COMPLETE PCB DESIGN TUTORIAL [2019] | ORCAD/ALLEGRO 17.2 HOW TO BUILD AND SIMULATE A SIMPLE CIRCUIT IN PSpice? | SRIKESH NAGOJI ORCAD HOW-TO SET UP CIS TUTORIAL CADENCE ORCAD ALLEGRO DOING PCB LAYOUT - LEARN ORCAD \u0026 CADENCE ALLEGRO ESSENTIALS (LESSON 9) INSTALLATION GUIDE OF ORCAD CADENCE v17.4 ELECTRICAL DESIGN SOFTWARE #ORCAD#CADENCE#INSTALL#GUIDE ORCAD 17.2 FULL INSTALLATION CADENCE ORCAD TUTORIAL - HOW TO GENERATE PCB (BOARD) IN ORCAD 17.4 INSTALL CADENCE SPB ALLEGRO AND ORCAD 2020 FULL WORKING 100% PRINTED CIRCUIT BOARD DESIGN : BEGINNER. STEP BY STEP MAKING OF PCBs AT HOME, DIY USING INEXPENSIVE MATERIALS ORCAD PSpice 16.6 TUTORIAL 0: DESCARGA E INSTALACI\u00c3\u00d3N DE ORCAD INTRO TO CADENCE 1: CREATING A SCHEMATIC AND SYMBOL HOW TO DOWNLOAD AND INSTALL L PSpice STUDENT VERSION 9.1L HOW TO START WITH CADENCE ALLEGRO - VERY SIMPLE TUTORIAL COMPLETE [ARDUINO UNO] SCHEMATIC IN ORCAD CAPTURE / ALLEGRO (2019) ORCAD TUTORIAL #1 - COME CREARE UNO SCHEMA ELETTRICO, ORCAD PCB DESIGNER 17.4

CADENCE ALLEGRO TUTORIAL - HOW TO CREATE SKILL SCRIPT AND YOUR OWN COMMANDS INSTALAR ORCAD - INSTALLING ORCAD PSpice CADENCE SPB ORCAD + CRACK [GOOGLE DRIVE+MEGA][INSTALL GUIDE] HOW TO INSTALL ORCAD 16.6 CADENCE ORCAD SIMPLE FLOW FROM SCHEMATIC TO PCB CADENCE ORCAD / ALLEGRO DESIGN ENTRY CIS DEMO TUTORIAL (PART 1) ALLEGRO ORCAD 17.2 INSTALLATION PROCESS | DOWNLOAD LINK ORCAD ALLEGRO HOW-TO CREATE COMPLEX FOOTPRINTS TUTORIAL ORCAD CADENCE ALLEGRO TUTORIAL CADENCE ORCAD AND ALLEGRO PCB EDITOR - CHANGE TEXT LINE THICKNESS ORCAD CAPTURE TUTORIAL IN TAMIL || SCHEMATIC DESIGN || CADENCE || NEWTECHNIFY

InstallScape is a Cadence application which facilitates the downloading and installation of Cadence software in a single process. The selected products can then be saved in a local Archive directory. Go to Downloads to obtain InstallScape, access whitepapers, user manuals, and more. Allegro FREE Physical Viewer

SOFTWARE DOWNLOADS - CADENCE

Customize Your PCB Design Experience with OrCAD new features like design reuse, advanced PCB routing technologies, In-design DFM, Interactive 3D Canvas

Tutorials | OrCAD

Quick Start Guide: Installing Cadence Allegro/OrCAD via CADpass R20 Prerequisites for Installing Allegro/OrCAD with CMC License Manager System (LMS). Have a unique login name (LINUX or...

System Requirements. One of the Cadence-supported operating systems is required. The following is recommended by ... *Tutorial paso a paso \u00c1 C\u00f3mo Instalar OrCAD 17.4 Full ...*

Cadence SPB/OrCAD 16.6 is a computer program. This page holds details on how to uninstall it from your PC. It was created for Windows by Cadence Design Systems, Inc.. Additional info about Cadence Design Systems, Inc. can be read here.

PCB Design Software | OrCAD | Cadence

One such tool from Cadence is the IR Drop Vision integrated analysis tool available in Allegro PCB Designer. This tool lets you see the power integrity status of your PCB design in real-time. It will show you pins that are not receiving a sufficient amount of voltage on a power net so that you can make immediate corrections to your layout.

Install Cadence OrCAD 17.4 : EMA Technical Support

Describes the design cycle for an electronic design, starting with capturing the electronic circuit in OrCAD Capture, simulating the design with PSpice, through the PCB layout stages in OrCAD Layout / OrCAD PCB Editor, and SPECCTRA, and finishing with the processing of the manufacturing output and maintaining the design through ECO cycles. To enable users to evaluate the power of the OrCAD PCB tools used in the Windows-based PCB design process.

Cadence Orcad Guide

Cadence Allegro and OrCAD products are integrated directly with Windows; the products support hardware and peripherals supported by Windows. A list of hardware and peripherals officially supported by Windows can be obtained from the Microsoft web page. The products require updating certain Microsoft libraries in the Windows directory.

Install OrCAD with Student Licensing : EMA Technical Support

About the Author Cadence PCB solutions is a complete front to back design tool to enable fast and efficient product creation. Cadence enables users accurately shorten design cycles to hand off to manufacturing through modern, IPC-2581 industry standard. Follow on LinkedIn Visit Website More Content by Cadence PCB Solutions

Quick Start Guide: Installing Cadence Allegro/OrCAD via ...

The "Cadence License Server Configuration Utility" is used to update your licensing for OrCAD products after you modify your purchase of OrCAD products or services you are licensed to use. If your updated license file is on a dongle, be sure the Cadence licensing dongle is plugged into your computer before you begin.

HOW TO USE MULTIPLE BATTERIES IN A CIRCUIT: A BEGINNER'S GUIDE

Starting with OrCAD and Cadence Allegro PCB - Tutorial for Beginners Installation guide for OrCAD/Allegro 2019 [[OrCAD 17.2 Tutorial] How to installing OrCAD 17.2 [full active] Tutorial Cadence HELP OrCAD and Allegro Complete PCB Design Tutorial [2019] | ORCAD/ALLEGRO 17.2 How to build and simulate a simple circuit in PSpice? | SriKesh Nagoji OrCAD How-To Set Up CIS Tutorial Cadence OrCAD Allegro Doing PCB Layout - Learn OrCAD \u0026 CADENCE ALLEGRO ESSENTIALS (LESSON 9) INSTALLATION GUIDE OF ORCAD CADENCE v17.4 ELECTRICAL DESIGN SOFTWARE #ORCAD#CADENCE#INSTALL#GUIDE OrCAD 17.2 Full Installation CADENCE ORCAD TUTORIAL - HOW TO GENERATE PCB (BOARD) IN ORCAD 17.4 INSTALL CADENCE SPB ALLEGRO AND ORCAD 2020 FULL WORKING 100% PRINTED CIRCUIT BOARD DESIGN : BEGINNER. STEP BY STEP MAKING OF PCBs AT HOME, DIY USING INEXPENSIVE MATERIALS ORCAD PSpice 16.6 TUTORIAL 0: DESCARGA E INSTALACI\u00c3\u00d3N DE ORCAD Intro to Cadence 1: Creating a Schematic and Symbol How to download and install l Pspice student version 9.1l HOW TO START WITH CADENCE ALLEGRO - VERY SIMPLE TUTORIAL COMPLETE [ARDUINO UNO] SCHEMATIC IN

OrCad Capture / Allegro (2019) OrCAD Tutorial #1 - Come creare uno schema elettrico, OrCAD PCB Designer 17.4

Cadence Allegro Tutorial - How to create SKILL Script and your own Commands [Instalar OrCAD - Installing OrCAD](#) Pspice-Cadence SPB-OrCAD + Crack [Google Drive+Mega][Install Guide] [How To Install Orcad 16.6 Cadence OrCAD simple flow from schematic to PCB Cadence OrCAD / Allegro Design Entry CIS Demo Tutorial \(Part 1\) Allegro-orCAD 17-2 Installation process | Download Link OrCAD Allegro How To create complex footprints Tutorial OrCAD Cadence Allegro Tutorial Cadence OrCAD and Allegro PCB Editor - Change Text Line Thickness OrCAD Capture Tutorial in Tamil || Schematic Design || Cadence || NewTechnify](#)

[32 Bit Microcontroller: A Beginner's Guide - Cadence Blog](#)

The "Cadence License Server Configuration Utility" is used to update your licensing for OrCAD products after you modify either your purchase of OrCAD products or the services you are licensed to use. Open your Windows Start menu and select Cadence > License Server Configuration Utility to launch the utility.

Optimization Guide for PCB Routing Constraints - Cadence Blog

YOUR PDN DESIGN GUIDE - CADENCE DESIGN SYSTEMS

The Cadence® OrCAD® /Allegro® FREE Physical Viewer is a free download that allows you to view and plot databases from OrCAD PCB Editor, Allegro PCB Editor, Allegro Package Designer, and Allegro PCB SI technology.

Related with Cadence Orcad Guide:

[© Cadence Orcad Guide Parallel Lines With Transversal Worksheet](#)

[© Cadence Orcad Guide Parallel Lines And Transversals Pyramid Puzzle Worksheet Answer Key](#)

[© Cadence Orcad Guide Parallel Circuit Practice Problems](#)

CADENCE ALLEGRO AND ORCAD 17.2-2016 INSTALLATION GUIDE FOR ...

Before you begin Welcome to OrCAD OrCAD offers a total solution for your core design tasks: schematic- and VHDL-based design entry; FPGA and CPLD design synthesis; digital, analog, and mixed-signal simulation; and printed circuit board layout. What's more, OrCAD's products are a suite of applications built around an engineer's design flow—not just a collection of independently developed ...

CADENCE SPB/ORCAD 16.6 VERSION 16.60.062 BY CADENCE DESIGN ...

OrCAD PCB designer, by Cadence, allows you to navigate through these challenges by setting and visualizing constraints in real-time, as well as saving constraints for future projects to avoid starting from scratch. If you're looking to learn more about how Cadence has the solution for you, talk to us and our team of experts. About the Author

[LAYARUG.PDF - \u00ae Layout Orcad Autorouter User\u2019s ...](#)

Trademarks: Trademarks and service marks of Cadence Design Systems, Inc. contained in this document are attributed to Cadence with the appropriate symbol. For queries regarding

Cadence's trademarks, contact the corporate legal department at the address shown above or call 800.862.4522.

Archivos(Orcad+Crack): <http://raboninco.com/oe1> (1 solo

acortador de enlace)AllegroCrackMaster(Limpiador):

<http://raboninco.com/10hcf> (1 solo acortador de ...