3d pdf to step converter

Trang 2 Trang 3 Trang 4 Trang 5 Trang 6 Trang 7 Trang 8 Trang 9 Trang 10 Trang 11 Trang 12 Trang 13 Trang 14 Trang 15 Trang 16 Trang 17 Trang 18 Trang 19 Trang 20 Trang 21 Trang 22 Trang 23 Trang 24 Trang 25 Trang 26 3D-Tool Phiên bản 14Free TrialDownload Giúp làm cho3D-Tool betterFeedback Trình xem CAD cho phép xác nhận trực quan các mô hình 3D trong tất cả các lĩnh vực của công ty mà không cần kiến thức CAD. 3D-Viewer / 2D-Viewer Mặt cắt ngang, lắp ráp explode, hoạt hình, PMI Hiển thị native 3D-CAD-File Định dạng CATIA, Pro / E, Creo, Inventor, UG Siemens NX, SolidWorks, SolidEdge, Rhino 3D-File Định dạng JT, DWG, DXF , STEP, IGS, VDA, SAT, X_T, X_B, STL, VRML, PLY, 3DS, XGL, OBJ 2D-File Format CATDrawing, SLDDRW, DXF, DWG, HPGL See all 3D-Tool file formats that provide practical tools for evaluating 3D design, including extensive measurement and analytical functions for tool manufacturing 3D measurement distance, angle, radius, edge, wall thickness, clearance, surface, 2D point-measuring distance, angle, radius, line length of draft analysis, undercuts, expected areas, thick walls, patterns comparing information volume, surface, weight view all features 3D Premium Tool simplifies the processing of native CAD 3D data from customers by converting them into popular 3D formats. Switch from: CATIA, Pro/E, Creo, Inventor, UG Siemens NX, SolidWorks, SolidEdge, Rhino, JT, DWG, DXF, STEP, IGES, VDA, SAT, Parasolid Convert into: CATIA V4/V5, STP, IGS, VDA, SAT, X_T and STL 3D-NativeCAD Converter I use 3D-Tool to collaborate with suppliers and customers Especially exporting an exe file with custom views and notes is very useful... Erik Raun Jensen, Danfoss Electronic Controls & Sensors, Danish Domestic, National and International Cooperation is critical to developing successful products. 3D-Tool facilitates quick, efficient, detailed cooperation... Paul Schwendinger, Honeywell, USA On 3D - Tools - we believe it's a great product and I've recommended it to our business partners... Constantin Tiba, Presstran Industries Cosma International, USA More testimonials Collaborate with your customers, suppliers, and colleagues in a multi-CAD environment. CAD Exchanger GUIView, analyzes and converts 3D data on desktopScatalys learn moreCAD Exchanger CloudSecurely view, share and communicate through the browser Learn moreDevelop 3D applications for desktop, web and mobile CAD Exchanger SDK, Web Toolkit and CLISoftware libraries for C#, C++, Java, Javascript and PythonBrowse development toolsThe industry needs to use casesDesign solutions for AR/VR, technical simulation, on-demand manufacturing and more. Discover frequent use casesSTEP or Standard to exchange product model data also known as ISO 10303. STEP was originally designed with the idea of replacing the IGES format (this is a neutral CAD file format first widely used by the supplier). However even today both coexisting formats and data in IGES and STEP represent lion sharing of all CAD files. data modeling and other CAx systems. STEP solves product data from mechanical and electrical design, shape and tolerability, analysis and production, as well as specific additional information for various industries such as automotive, aerospace, construction, construction, ships, oil and gas, processing plants and other industries. STEP is developed and maintained by the ISO technical committee. There are also different technical groups that define usage conventions and recommended practices to ensure better interaction between software applications. In cad world, the most prominent group is www.cax-if.org with software providerss and industrial users. ISO10303 is very large and complex and is structured in several layers, for example, from defining physical file formats (Part 21) to defining external squares/topology (Part 42), up to application protocols (AP) that define higher-level industry use cases. CAD Exchanger supports AP203, A214, and AP242, ap's most commonly used. Convert STEP (.stp, .step) to ACIS (.sat, .sab)Convert STEP (.stp, .step) to Open CASCADE (.brep)Convert STEP (.stp, .step) to Collada (.dae)Convert STEP (.stp, .step) to DXF (.dxf)Convert STEP (.stp, .step) to FBX (.fbx)Convert STEP (.stp, .step) .step) to GLTF (.gltf, .glb)Convert STEP (.stp, .step) to IFC (.ifc)Convert STEP (.stp, .step) to IGES (.igs, .iges)Convert STEP (.stp, .step) to JT (.jt Convert STEP (.stp, .step) to OBJ (.obj)Convert STEP (.stp, .step) to Rhino (.3dm)Convert STEP (.stp, .step) to STL (.stl)Convert STEP (.stp, .step) to VRML (.wrl). Convert STEP (.stp, .step) to X3D (.x3d)Convert STEP (.stp, .step) to Parasolid (.x_t, .x_b, .xmt_txt, .xmt_bin, .xmp_txt, .xmp_bin)Convert 3DS (.3ds) to STEP (.stp, .step)Convert Open CASCADE (.brep) .3mf to STEP (.stp, .step)Convert CATIA (. CATPart, . CATProduct) to STEP (.stp, .step)Convert Collada (.dae) to STEP (.stp, .step)Convert PTC Creo (.prt, CATProduct) to STEP (.stp, .step)Convert PTC Creo (.prt, .asm) to STEP (.stp, .step)Convert DWG (.dwg) to STEP (.stp, .step)Convert DXF (.dxf) to STEP (.stp, .step)Convert FBX (.fbx) to STEP (.stp, .step)Convert IFC (.ifc) to STEP (.stp, .step)Convert IGES (.igs, .iges) to STEP (.stp, .iges) to STEP (.stp, .iges) to STEP (.stp, .step)Convert JT (.jt) to STEP (.stp, .step)Convert Siemens NX (.prt) to STEP (.stp, .step)Convert OBJ (.obj) to STEP (.stp, .step)Convert Rhino (.3dm) to STEP (.stp, .step)Convert Solidworks (.sldprt, .sld)Convert Solidworks (.sldprt, .sldasm) to STEP (.stp, .step)Convert STL (.stl) to STEP (.stp , .step)Convert U3D (.u3d) to STEP (.stp, .step)Convert VRML (.wrl) to STEP (.stp, .step)Convert X3D (.x3d) to STEP (.stp) to STEP (.stp), .step)Convert Parasolid (.x_t, .x_b, .xmt_txt, .xmt_bin, .xmp_txt, .xmp_bin) to STEP .step) Professional 3D Viewer to read DWG, STEP/STP, STL, IGES/IGS, SLDPRT, X_T, X_B and other 3D file formats. The 3D viewer can print and convert 3D files. If your download does not start automatically click here. You'll also receive an email with a download link. Video Tutorial The main technique CAD Guide Easily receive STEP help (or 'ISO 10303', Standard for Product Data Exchange) is a complex set of specifications that define data exchange standards for a wide range of industries and applications. Each is identified, maintained and ratified by various ISO technical committees. STEP AP203 & AP214 is the CAD provider's primary neutral 3D data translation file format designed for high-definition data exchange between all major professional MCAD & CAD 3D model applications. They have grown as a successor to file format neutrality with IGES providers but today you will find STEP and IGES data files coexisting together. STEP files consist of multiple parts, which are approved specifications (documents) according to their own rights. The term part should not be confused with the common form of part file terminology used in the MCAD industry (where according to which most MCAD files are defined as assembly files or 3D part files). STEP sections (document specifications) range from high-level application protocols (such as AP203, AP214, AP242) to low-level generic resouces such as #41 and materials (#45). Step files from the CAD and MCAD industries will often contain basic elements and structures: NURBS analysis/parameters of SCBS/bicubic curves & 3D polylines 'BREP' topology information (to determine 'solids' polymers) CAD assembly hierarchy and instancing instancing instancing meta-unit-of-measure data naming parts, tolerances and other pointer information source-modeller And basic materials. Most people originally came to Okino asking for a STEP importer. STEP is a very good and stable MCAD conversion file format but it should only be used in specific situations as outlined in our CAD Data Supply Rules and Proposals. If you do not have access to the original CAD file then STEP AP214 will be the best file format to request from your data provider, or a properly exported IGES BREP solids file. You'll want to request that the step files be exported as 'BREP solids' (and not 'surfaces') and in the AP214 STEP variant. Okino of PolyTrans| CAD provides the actual STEP 3D file import conversion solution used by major engineering industries & Professional, aerospace, military, enterprise, animation/multimedia and VR/AR. Okino has been at the fore in the MCAD conversion market for over 30 years and as such our step entry solution has been designed to handle super large data sets that would otherwise suffocate and pull other conversion solutions to their knees. Okino software is well known and sold especially, because of its ability to process step files from smallest to largest in a solid and powerful + efficient way. Cp3 scooter concept image. Switch via Okino's PolyTrans| CAD system. Christoph Bodensieck, © Industrial PDD & Switch Mobility. Click on the image to see the full-scale gig. Engineering About Okino's STEP importer can be read here. Okino also recommends importing your CAD files through these native importers (among others): The first question we ask our new customers is 'Where does the MCAD program come from?'. That information can be very important, depending on the name of the source MCAD program. Step files can easily read ascii text files that can be opened in any text editor in general. After you open a file in such a text editor, you'll want to find text labeled 'FILE_NAME' close to the beginning of the file. The name of the originating MCAD program will usually be identified in that subsc section. Here are some good examples: FILE_NAME ('FILENAME. STEP', 'T08:28:23', (), (), 'SwSTEP 2.0', 'SolidWorks', ''); FILE_NAME('FILENAME.stp', 'T02:22:23',(), (), 'Inventor,'Autodesk Inventor','); FILE_NAME('FILE NAME', 'T23:57:22',(),(),''), 'PTC', 'CREO PARAMETRIC','); FILE_NAME(/* name */ 'FILENAME', /* time_stamp */ 'T18:05:02+01:00', /*author*/ ('My Name'), /* organization*/ ('Company Name'), /* preprocessor_version*/ 'ST-DEVELOPER v10', /* originating_system*/ 'UGS - NX', /* authorization *' The top-level parts of the STEP file are called application protocols (APs). These application protocols then refer to the general parts of ISO 10303 to meet data exchange requirements specific to a particular application or industry. For the benefit of Okino customers, the popular AP used to transmit MCAD 3D data is the newer AP203, AP214 and AP242 (with AP203 and AP214 being the most common). AP203 (and parts 1, 11, 21, 31, 41, 42, 43, 44, 46 and 101) was first published by ISO in 1994. Its official title is the 3D design control configuration of mechanical parts and ass councils. AP203 remains a fundamental aspect of step file import and export in many MCAD progrms. AP214 is the second phase of STEP's development (ended in 2002) serving the aerospace, automotive, electrical, electronics and other industries. Its official title is the core data for automotive mechanical design processes. AP214 is essentically the same as 'AP203 Edition #2' in the MCAD domain. AP242 was created to overcome some of the great causes of AP204 and AP214, notably their large APIs, which overlap too much between APIs and lack harmony. The first edition was published by ISO in December 2014. Its official name is Managed model based on 3D technique. AP242 was created by merging the top-level protocols of AP203 (aerospace) and AP214 (automotive). AP242 is also added in the ability to express simple tessellated shapes. AP203 (3D design control configuration of mechanical parts and assembly) is a core and fundamental aspect of STEP file format specification used in the MCAD and CAD industries. Core capabilities including: Products are mechanical parts and assembly; Five types of shape performers: wire frame and no surface wireframes with link structures, diverse surfaces with connected structures, face boundary representations, and boundary representations. Editition #2 (2011) combines the main aspects of AP214, in partcular color and layering ability. AP214 (Core Data for Automotive Mechanical Design Processes) is another core STEP file format variant used in the MCAD and CAD industries. According to its official definition: AP214 describes an application reference environment for creating and exchanging volume-based design data during computer-aided mechanical design, along with appropriate data models and a form of physical file implementation. The information model supports all the physical and outer tocribing aspects of a full description of the shape and size of an object. It was originally developed for applications in mechanical engineering design using CAD modeling techniques that model solid boundary representation (B-rep) and may be suitable for other equipment fields using this technique. Core capabilities include: Three types of B-rep models used to represent shapes: B-rep facetted models; B-rep model with basic surface; B-rep model with sculpted surface; Curves and surface shapes; Curves are defined in the parameters space (pcurves); Diverse distributions; Units and measures related to the elements of theeology; Council of departments and sub-councils. AP242 (Managed model based 3D engineering) was created by merging the top-level protocols of AP203 (aerospace) and AP214 (automotive). The first edition was published by ISO in December 2014 with the goal of successfully merging and replacing AP203 and AP214, as well as expanding them with some necessary additional capabilties. Version 2 (in progress) is an extension to the electrical design domain with specific improvements to PDM, 3D photoology, 3D PMI, composite and mechanical design. Another goal of AP242 is long-term CAD data storage. It is of strategic importance to the aerospace and defense industries, and to the automotive industry. AP242 is added in the ability to express simple tessellated shapes. It is also designed as a modular file format to overcome some of the great exatheses of AP204 and AP214, notably their large APIs, which overlap too much between APIs and lack harmony. STEP files are usually easy to read, usually with one version per line. This section will provide a very rough overview of how to visually read ascii content of a STEP file. Let it first be said that the STEP files are too secreted and large. For example, the definition of 3D peak coordinates is simple (-2.0, 0.9, 0.8) occupy the entire line in the file as follows (and many STEP files can have hundreds of thousands to millions of these basic cyetic coordinates): #48 = CARTESIAN_POINT ('Ctrl Pts',(-2.0, 0.93687370014661, 0.8185109331703)); For this example, we will check the determination of a simple syceology model created by Inventor. In some 3D file formats, the definition of a 50-year-old may be as simple as a line, but STEP requires 206 lines. In short, we'll provide a full list of STEP files in a separate HTML page for your review. From the original step file, we chose to break, simplify and rearrange part of it that defines only the BREP link structure of the primitive 500. Rearranging the lines of the file was done in the following hierarchy, where each line referenced to a sub-family from another line. #12=ADVANCED_BREP_SHAPE_REPRESENTATION(', (#14),#108); #14=MANIFOLD_SOLID_BREP ('Solid1',#25); #25=CLOSED_SHELL(',(#24)); #24=ADVANCED_FACE(' ',(#15), #23.T.); #15=FACE_OUTER_BOUND(' ', #16,.T.); #16=EDGE_LOOP(', (#21.#22)); #21=ORIENTED_EDGE(',*,*, #20,.T.); #22=ORIENTED_EDGE(',*,*, #20,.F.); #20=EDGE_CURVE('',#18, #19.#17.T.); #17 = B_SPLINE_CURVE_WITH_KNOTS (' , ', 3,(#99, #100, ... Etc... #18=VERTEX_POINT('', #97); ... Etc... #19=VERTEX_POINT(', #98); ... Etc... #23=B_SPLINE_SURFACE_WITH_KNOTS('',3, 3,((#48, #49,... #48=CARTESIAN_POINT('Ctrl Pts', (-2., -0.06, -0.1)); ... Etc... In this way, the STEP file and its file hierarchy are much easier to understand. Each version is defined by a unique identifieer, starting with '#' (such as '#12'). The name of the unique file data type is written in capital letters, such as ADVANCED_BREP_SHAPE_REPRESENTATION. Each instance references one or more child bodies using their unique version identifies. Okino of PolyTrans| CAD intelligently imports, optimizes, and converts STEP files into all major 3D file formats, animation packages, military VisSim, virtual reality (VR) and augded reality (AR) systems (such as Unity and Unreal Engine) and third-party/OEM integration. Step parts and assembly files can be easily and directly converted into popular & popular destinations such as STEP (.step, .stp) to AutoCAD (.dwg, .dxf) STEP (.step, .stp) to 3ds Max (.max) STEP (.step, .stp) to CADMATIC (.3dp) STEP (.step, .stp) to CINEMA 4D (.c4d) STEP (.step, .stp) to COLLADA (.dae) STEP (.step, .stp) to DWF-3D (.dwf) STEP (.step, .stp) to FBX (for Unity, Unreal and MODO) STEP (.step, .stp) to JT Open (.jt) STEP (.step, .stp) to LightWave (.lws, .lwo) STEP (.step, .stp) to Maya (.mb) STEP (.step, .s)tp) to Microstation DGN (.dgn) STEP (.step, .step, .stp) to Microstation DGN (.dgn) STEP (.step, .step, .stp) stp) to OpenFlight (.flt) STEP (.step , .stp) to NGRAIN 3KO (.3KO) STEP (.step, .stp) to Rhino-3D (.3dm) STEP (.step, .stp) to SketchUp (.skp) STEP (.step, .stp) to STL (.stl) STEP (.step, .stp) to Visual Components STEP (.step, .stp) to Acrobat-3D (PDF via U3D) (.u3d) STEP (.step, .stp) to VRML2 & X3D (.wrl, .x3d) STEP (.step, .stp) to Wavefront (.obj) STEP (.step, .stp) to dozens of kemppi 3D file format welding machines, STEP to LightWave, by Spikey Animation and Ice Cream Watch explained here. It's here.