

SOLIDWORKS Student Edition FAQs

Question: Where can I download the SOLIDWORKS Student Edition?

Answer: You can download the Student Edition from www.solidworks.com/SEK

If you have a serial number (9020...), you can follow these steps to download:

1. Go to the link www.solidworks.com/SEK
2. Select **"Yes"** for the Option "I already have a serial number that starts with 9020"
3. Choose the appropriate SOLIDWORKS version from the drop/down list
4. During the download you will need your serial number to install and activate the license

If you have an SDK-ID:

1. Go to the link www.solidworks.com/SEK
2. Select **"No"** for the Option "I already have a serial number that starts with 9020"
(Note: Your serial numbers will be displayed on screen later, as well as provided via email, once you proceed for downloading)
3. Enter your **SDK-ID** (This ID can be obtained from your teacher/professor at school/college/university)
4. Choose the appropriate SOLIDWORKS version from the drop/down list
5. During the download you will need your serial number to install and activate the license

Question: I did not receive an email with my serial number after requesting the download for the Student Design Kit (SDK)? What should I do?

Answer: Please check the following:

1. If you did not receive a serial number via email, check your **spam/junk** folder as the email might have been filtered
2. When you request a download for a Student Design Kit (SDK) using your SDK-ID, serial numbers are displayed on the download page. Copy the serial numbers and use them during the installation

Question: I purchased a Student Edition a week ago but I have not yet received an email with serial numbers for installation? What should I do?

Answer: If you did not receive serial numbers via email, check your **spam/junk** folder as the email might have been filtered. You may also contact the online store/reseller from whom you purchased the Student Edition and provide them with your order number to get more information.

Question: Why do I get an error for invalid SDK-ID when I am trying to download SOLIDWORKS?

Answer: It indicates that the SDK-ID has already expired. An SDK-ID is sent to a school/college/university from the reseller. Please contact your school/college/university to get the newest SDK-ID. If the SDK-ID has expired, the school needs to contact their reseller to request for a new ID.

Question: I am using the SOLIDWORKS Student Edition. Do I have access to SOLIDWORKS Knowledge Base articles?

Answer: Yes, students with Student Edition and SEK serial numbers have access to the MySolidWorks.com portal, through which they can access the SOLIDWORKS Knowledge Base. Please note that Student Design Kit (SDK) users do not have access to the MySolidWorks web portal, except for limited “Guest” access. More information is available here: <https://my.solidworks.com/account/student-access>

Question: Can students get access to the MySolidWorks web portal?

Answer: Yes, students with a valid serial number for Student Edition and Student Engineering Kit (SEK) products can access the MySolidWorks.com web portal. Please note that the Student Design Kit (SDK) does not provide access to the MySolidWorks web portal, except for limited “Guest” access. More information is available here: <https://my.solidworks.com/account/student-access>

Link to MySolidWorks : <https://my.solidworks.com/?&lang=en>

Steps to register: <https://my.solidworks.com/account/student-registration-direction>

Question: I am unable to create a Customer Portal account using my Student Edition serial number. What should I do?

Answer: A Customer Portal account cannot be created using a Student Edition serial number. It will only accept commercial license numbers. However, Student Edition serial numbers have access to the MySolidWorks portal. Please refer the above questions for information on the MySolidWorks portal.

Question: I have created my Customer Portal account but the Download is locked. How can I access the download?

Since the customer portal account is for commercial users, the download is locked when logging in using a “Guest” account.

Please refer to above FAQ answers to get more information about how to download the Student Edition.

Question: I have a Student Edition serial number; can I upgrade to the latest version or can I download & install the latest version?

Answer: No, Student Edition serial numbers are version specific. Hence, it is not possible to install other versions with the same serial number. You can only install the version entitled by your serial number.

Question: I have lost my serial numbers for my Student Edition. How can I get my serial numbers again?

Answer: Please contact the online store/reseller from whom you have purchased the Student Edition license. They can provide you with information regarding your serial numbers and/or any questions related to your order process.

Question: Why do I receive a warning, "Your serial number does not entitle you to SOLIDWORKS <version>"?

Answer: Student Edition serial numbers are **version-specific**. Hence, this warning will be given if the serial number used for installation is not entitled to the version selected during installation (mismatch). From the download drop/down list, make sure to select the entitled version (the version you purchased) and then download and install again.

Question: Can I install the SOLIDWORKS Student Edition 2017-2018 version using a serial number for Student Edition 2018-2019?

Answer: No. Although SOLIDWORKS is backward compatible, Student Edition serial numbers are **version-specific**. Hence, it is not possible to install Student Edition 2017-2018 using a Student Edition 2018-2019 serial number. Your newer license of SOLIDWORKS will, however, be able to read files created in older releases of SOLIDWORKS.

Question: Can I install my Student Edition on multiple computers?

Answer: Student Edition serial numbers allow for two simultaneous activations. You can install and activate SOLIDWORKS on two systems at a time. Hence you can use the same serial number to install and activate SOLIDWORKS on another system.

Question: My laptop hard drive died/crashed. I have replaced it. Can I install SOLIDWORKS again using the same serial number?

OR

I was unable to deactivate the license from my old computer. Can I install SOLIDWORKS on a new computer with the same serial number?

Answer: Student Edition products allow for 2 activations. If both activations have already been consumed, then you can contact our support team for further assistance by filling in the [Student support form](#). If you only installed and activated SOLIDWORKS once, then you can install and activate the license one more time without contacting SOLIDWORKS for assistance.

Question: How can I troubleshoot the error "Activation transaction failed: Empty response received from the Activation server"?

Answer: Please try the following to troubleshoot the issue:

1. Rename the FLEXnet folder and try the activation again:
Go to C:\ProgramData\FLEXnet and rename the folder FLEXnet (may be to FLEXnet_old)
(The ProgramData folder is a hidden folder. Either you can enable to show hidden folders from Folder options or you can directly type c:\Programdata into the address bar and press enter to access this folder. You will be able to find the Flexnet folder)
2. Try the activation again

Question: How do I troubleshoot the activation error 'Activation Wizard initialization issue code 72'?

Answer: This issue can occur if the SOLIDWORKS® License Service is not installed properly on the machine. In most of the cases, this issue is resolved by performing a repair installation of SOLIDWORKS. Follow the below steps for a repair installation:

1. Login as an administrator
2. Go to Start > Control Panel > Program and Features > SOLIDWORKS and click 'Change' to repair the installation
3. Select 'Repair the individual installation (on this computer)'
4. Click 'Next'
5. Follow the instructions to complete the modification

After the repair is successful, try following the steps to perform the activation:

1. Rename the FLEXnet folder and try activation again:
Go to C:\ProgramData\FLEXnet and rename the folder FLEXnet (may be to FLEXnet_old) (The ProgramData folder is a hidden folder. Either you can enable to show hidden folders from Folder options or you can directly type c:\Programdata into the address bar and press enter to access this folder. You will be able to find the Flexnet folder)
2. Try the activation again

Question: Is SOLIDWORKS supported on Mac OS? Can I install a Student Edition on Mac OS?

Answer: SOLIDWORKS is not officially supported in the Mac environment. We are aware of many users who run SOLIDWORKS successfully in a dual-boot Windows environment on Macs. We are not aware of many users running successfully in an emulated Windows environment on Macs. If you choose to run SOLIDWORKS on a Mac, your best chance for success will be choosing a dual-boot "Boot Camp" setup (booting in Windows) on your Mac.

SOLIDWORKS eDrawings and the free 2D CAD product DraftSight are supported in native Mac OS.

Please refer to this link providing system requirement information for installing SOLIDWORKS:

<http://www.solidworks.com/sw/support/SystemRequirements.html>

Question: Why does the following error message occur during download and install? "Could not connect to the SOLIDWORKS server to determine which products you are entitled to install. Check the Internet connection and click retry"

Answer: This error can occur if the Installation Manager is not able to connect to the SOLIDWORKS download servers. Try the following to resolve the issue:

Procedure A:

1. Set Internet Explorer as your default browser and reset Internet explorer to default settings. In the IE browser > Tools > Internet Options > Advanced and click "Restore advanced settings" and click "Reset" at bottom.
2. Close browser and restart it
3. Change IE compatibility mode: In IE menu >Tools > Compatibility View (select the option) and/or Compatibility View Settings, check the box "Display all websites in Compatibility View"
4. Try to download and install SOLIDWORKS

Procedure B:

If the issue is still not resolved, add the following domains to the firewall's/proxy server exceptions list:

1. im.solidworks.com
2. im-ak.solidworks.com
3. dl-ak.solidworks.com

use port 80 for adding all the above exceptions. Then try to download and install again

Procedure C:

There might be an issue with your Internet connection. Please check with your Internet service provider and see if they can help.

Question: When I run the installation, Installation Manager is blank. Why?

Answer: When the installation files are downloaded and extracted it is possible that Windows will block the HTML files and scripting that creates the user interface for the Installation Manager. This results in the Installation Manager not displaying any options and showing a white screen.

To correct this issue, the files for the Installation Manager will need to be unblocked.

1. Navigate to the <SOLIDWORKS Download>\sldim\lang\english\HTML folder that was extracted from the download
2. Right click on a file found in the folder and select Properties
3. On the General tab select the 'Unblock' option
4. Do this for every file in this folder. The list of files should include the following:
 - a. frameset.html
 - b. framesetstartup.html
 - c. header.html
 - d. imfunc.js
 - e. navpanestartup.html
 - f. sldim.css
 - g. sldim_navepane.css

Question: How can I troubleshoot the error "Additional files are needed based on the products selected..." during installation of Student Edition?

Answer: Student Edition serial numbers are version-specific. This error can occur if a previous version serial number was used for installing a new version. (i.e., the serial number used for the installation is not entitled to the version you are trying to install).

It might be possible that the previous version serial number was automatically taken from the registry (as it was available in the registries) on the serial number page and the user continued without modifying it.

Hence, on the Serial Number page of the Installation Manager, please ensure that the correct serial number is entered. If any serial number is automatically displayed, remove it and enter the correct serial number. Proceed with the installation.

Question: Why does the installation of SOLIDWORKS generate a message that it is not supported on Windows® XP, when the installation is being installed on a PC that is running Windows® 7 x64 or Windows® 10 x64?

Answer: Usually when the Installation Manager (IM) is reporting that incorrect operating system is in use, it indicates that the SOLIDWORKS Installation Manager (IM) is running in "compatibility mode."

Please try the following:

There are 3 possible .exe files that could cause this.

1. SolidWorksSetup.exe (the file that is downloaded),
2. setup.exe (launches the IM),
3. sldim.exe (the actual installation manager)

Right click on each .exe and select Properties, then select -> Compatibility tab; ensure that it is not set to "run as XP." If it is, unselect this option and then re-run the installation.

Question: Error "The Student Edition has expired. Please browse to the following web location to find out about the new version of SOLIDWORKS Student Edition: www.solidworks.com/goEDU."

How can I troubleshoot the above error when starting SOLIDWORKS after a successful installation?

Answer: This error can occur if prior license information is still available on your system, or if the installation is performed with a prior version serial number.

Try the following to troubleshoot:

1. Rename the FLEXnet folder and try the activation again:
Go to C:\ProgramData\FLEXnet and rename the folder FLEXnet (may be to FLEXnet_old) (The ProgramData folder is a hidden folder. Either you can enable to show hidden folders from Folder options, or you can directly type c:\Programdata into the address bar and press enter to access this folder. You will be able to find the Flexnet folder.)
2. Try to activate again and if the error still occurs, confirm that a valid serial number is used for the SOLIDWORKS installation by modifying the installation. If the serial number is not a valid one, please change and input the correct serial number:
 - a. Go to Start > Control Panel >SOLIDWORKS
 - b. click 'Change' to modify the installation. Select 'Modify the individual installation (on this computer)' >click 'Next'
 - c. Under 'serial number' make sure that your valid serial number is in use
 - d. If not, replace the serial number with a valid one and click 'Next'
 - e. Follow the instructions to complete the modification
 - f. Rename the FLEXnet folder and try to activate again

If the problem still occurs, try doing a repair installation of SOLIDWORKS by following the steps below:

1. Go to Start > User Account > User Account Control Settings. Set it to "Never Notify". Restart the system
2. Login as administrator
3. Turn off all anti-virus, popup blocker, ad-ware and spy-ware applications along with Windows Firewall
4. Go to Start > Control Panel >SOLIDWORKS and click 'Change' to repair the installation
5. Select 'Repair the individual installation (on this computer)'
6. Click 'Next'. Follow the instruction to complete the modification
7. Try activating again

**Question: While installing, I get one of the error messages below: "Product:SOLIDWORKS ErrorCode:1722
CA:register_i386_SldShellExtServerExe Location:C:\Program Files\SOLIDWORKS
Corp\SOLIDWORKS\sldShellExtServer.exe Command:sldshellserver.exe /REGSERVER"**

OR

**Internal error: The Windows Installer for this product component did not run as expected:
Register_DocMgrDLL.9FB07B3D_0B7D_4A2F_82AB_F16B1F24912C.: Contact Technical Support.}**

How to I resolve it?

Answer: This error is related to the installation of prerequisite VC++ components. To troubleshoot the issue, try reinstallation of VC++ components manually:

1. Go to Start > User Account > User Account Control Settings. Set it to "Never Notify". Restart the system
2. Login as administrator
3. Uninstall VC++ redistributable components (2008 and 2010 and 2012, 2013, 2015) from Programs and Features
4. Restart the computer
5. Login as administrator
6. Reinstall: VC++ 2008 and 2010 and 2012, 2013, 2015
 - [a] downloaded files\prereqs\VCRedist9\vc_redist_x86.exe and vc_redist_x64.exe
 - [b] downloaded files\prereqs\VCRedist10\vc_redist_x86.exe and vc_redist_x64.exe
 - [c] downloaded files\prereqs\VCRedist11\vc_redist_x86.exe and vc_redist_x64.exe
 - [d] downloaded files\prereqs\VCRedist12\vc_redist_x86.exe and vc_redist_x64.exe
 - [e] downloaded files\prereqs\VCRedist14\vc_redist_x86.exe and vc_redist_x64.exe
7. Restart your computer
8. Login as administrator
9. Install SOLIDWORKS

Question: After a successful installation of SOLIDWORKS, when I double click the desktop icon to start SOLIDWORKS nothing happens. How can I troubleshoot this problem?

Answer: Generally, this issue can be seen if prerequisites are not installed correctly.

To verify this: Try running SLDWORKS.exe from the install directory (Default location is "C:\Program Files\SOLIDWORKS Corp\SOLIDWORKS") and see what happens. Do you see "Side by Side" error message window showing: "The application has failed to start because its side-by-side configuration is incorrect...."

If yes, it points to an issue with the installation Microsoft Prerequisites "Microsoft Visual C++ Redistributable" required for SOLIDWORKS. Please try the following steps to troubleshoot the issue:

1. Go to Start > User Account > User Account Control Settings. Set it to "Never Notify". Restart the system
2. Login as administrator
3. Uninstall VC++ redistributable components (2008 and 2010 and 2012, 2013, 2015) from Programs and Features
4. Restart the computer
5. Login as administrator
6. Reinstall: VC++ 2008 and 2010 and 2012, 2013, 2015
 - [a] downloaded files\prereqs\VCRedist9\vc_redist_x86.exe and vc_redist_x64.exe
 - [b] downloaded files\prereqs\VCRedist10\vc_redist_x86.exe and vc_redist_x64.exe
 - [c] downloaded files\prereqs\VCRedist11\vc_redist_x86.exe and vc_redist_x64.exe
 - [d] downloaded files\prereqs\VCRedist12\vc_redist_x86.exe and vc_redist_x64.exe
 - [e] downloaded files\prereqs\VCRedist14\vc_redist_x86.exe and vc_redist_x64.exe
7. Restart computer
8. Login as administrator
9. Install SOLIDWORKS

If the problem still occurs, repair the installation of SOLIDWORKS:

1. Go to Start > User Account > User Account Control Settings. Set it to "Never Notify". Restart the system
2. Login as administrator
3. Turn off all anti-virus, popup blocker, ad-ware and spy-ware applications along with Windows Firewall
4. Go to Start > Control Panel > SOLIDWORKS and click 'Change' to repair the installation
5. Select 'Repair the individual installation (on this computer)'
6. Click 'Next'
7. Follow the instruction to complete the modification

Question: When I install the SOLIDWORKS® software, why does the following warning appear "A Windows reboot is pending from a previous installation..."?

Answer: To help assure a successful installation of the SOLIDWORKS® software, the Installation Manager checks to see if there are any pending reboot requests on your system. A pending reboot request occurs when a program or installation makes a change to files, registry keys, services, or operating system settings. This leaves the system in a transient state.

For example, a SQL Express installation automatically terminates if a pending reboot request exists. In other cases, the transient state of a SOLIDWORKS installation can lead to installation errors.

As a best practice, restart your system before beginning a SOLIDWORKS installation. If it is not possible to restart the computer at the time of install, then you can ignore the warning. In such cases, restarting the system becomes the first troubleshooting step following an unsuccessful installation

Question: The Installation Manager shows an error related to SOLIDWORKS Electrical when I am trying to install. How do I resolve this error?

Answer: This error is related to the installation of SQL Server, which is required for the SOLIDWORKS Electrical software included in your package.

Do you use SOLIDWORKS Electrical? If not, you can deselect it for installation. To do that, run the installation again and on the summary page, click change for 'Product Selection,' deselect SOLIDWORKS Electrical from the list, and proceed with the installation.

If you use SOLIDWORKS Electrical, please try the following to troubleshoot the issue:

Under Electrical Options, on the Summary page of the Installation Manager, make sure the correct SQL server instance is selected in the Electrical options of the Installation Manager and try the installation.

You can try the following:

- Try renaming the current instance TEW_SQLEXPRESS to TEW_SQLEXPRESS_1
- Try to install with "Install with a new instance of SQL Server"

Question: I am unable to uninstall SOLIDWORKS from Control Panel > Program and Features; uninstallation fails with errors. How can I uninstall it to install the new version of SOLIDWORKS?

Answer: You can contact [SOLIDWORKS Student Edition Support](#) and provide the Installation Manager logs.

To go to the Installation log folder, you can type %appdata% at Start > RUN and press Enter, and it will open the "Roaming" folder. In the "Roaming" folder, go to the "SOLIDWORKS" folder. Under the "SOLIDWORKS" folder, you will find a folder named "Installation log." Zip this complete folder and provide it for investigation.

However, before contacting the support team, you can follow this procedure to troubleshoot the issue:

Try a complete uninstallation of SOLIDWORKS using the Microsoft fixit tool:

1. Download the Microsoft fixit tool from this link:
<https://support.microsoft.com/en-us/help/17588/fix-problems-that-block-programs-from-being-installed-or-removed>
2. Run the downloaded tool and click Next
3. Click Uninstalling
4. In product selection select ' SOLIDWORKS (version) ' and click Next to follow uninstallation process
5. Repeat the process until all the versions of SOLIDWORKS and all the SOLIDWORKS products are uninstalled
6. Try installation of SOLIDWORKS

Question: SOLIDWORKS crashes on startup.

Answer: In most of the cases this problem is caused by unsupported graphics cards and drivers. To verify this, try to start SOLIDWORKS with Software OpenGL. Since SOLIDWORKS is not starting at all in this scenario, you can try to start SOLIDWORKS with Software OpenGL through SOLIDWORKS Rx:

1. Launch SOLIDWORKS Rx from Start > All Programs > SOLIDWORKS(version) > SOLIDWORKS Tools
2. On the 'Home' tab of Rx under 'SOLIDWORKS Safe Modes' click the option "Click here to launch SOLIDWORKS in Software OpenGL mode."

If SOLIDWORKS does not crash in OpenGL mode, this confirms that the problem is due to the graphics card or its drivers. To see the list of SOLIDWORKS supported graphics cards and drivers for your system (if your system is not listed please select Any System Vendor) follow the link: <http://www.solidworks.com/sw/support/videocardtesting.html>

Question: SOLIDWORKS crashes when trying to create a new part, assembly, drawing, or trying to open a saved part.

Answer: In most of the cases this problem is caused by unsupported graphics cards and drivers. To verify this, try to start SOLIDWORKS with Software OpenGL (if this option is not disabled due to unsupported graphics card). You can either follow the steps in the above question, or follow the steps below:

1. Launch SOLIDWORKS and without opening any document go to Tools > Options > Performance
2. Enable "Use Software OpenGL"
3. Select OK and now try to open a new part, assembly, or drawing

If SOLIDWORKS does not crash in Software OpenGL mode, this confirms that the problem is due to the graphics card or its drivers. To see the list of SOLIDWORKS supported graphics cards and drivers for your system (if your system is not listed please select Any System Vendor) follow the link: <http://www.solidworks.com/sw/support/videocardtesting.html>

Question: My Student Edition installation fails with "Dialog Shown: {0: The Installation Manager encountered a fatal error.: Check the log files and then contact Technical Support.}. How can I troubleshoot the error?

Generally, in this case, installation logs show an error with the installation of VSTA (vsta_aide.msi).

When trying to manually install the VSTA component by running vsta_aide.msi, installation of VSAT fails with error code 1305 "Error reading from file' during installation."

Answer: Generally, this occurs when the installation is running from OneDrive and the Installation Manager is unable to read some files from OneDrive leading to installation failure.

To resolve the issue, copy the all the downloaded files to a folder that's local to the system, then run the installation from the local folder.

Question: What can cause "Failed to initialize Visual Basic for Apps, equations and macros will not work. Are you low on disk space?" upon start-up of SOLIDWORKS®?

Answer: This problem can occur if users have insufficient permission/access to write to "HKEY_CURRENT_USER\Software\SOLIDWORKS" (registry key). Please ensure the user has write access to the above registry key. In conjunction, ensure the user has at a minimum read access to "HKEY_LOCAL_MACHINE\Software\SOLIDWORKS" as it populates the former registry key from this information. (you can run registry editor from Start > Run > type "regedit" (without quotes) and select OK. This will launch registry editor).

Caution:

Incorrectly editing the registry can severely damage your operating system. The SOLIDWORKS Technical Support team strongly suggests that you back up the registry data before making any changes to the registry. In addition to the above cause, this problem can also occur if there is problem with the installation of a Microsoft component (Microsoft Visual Basic 7.1) required for SOLIDWORKS.

If the above solution does not work, please try the following to troubleshoot the issue:

- 1) Uninstall Microsoft Visual Basic 7.1. This program is not visible in Programs and Features, so open a command prompt and type: `msiexec.exe /x{90120064-0070-0000-0000-4000000FF1CE}`
- 2) Restart the computer
- 3) Browse to <DVD or SOLIDWORKS Download location>\prereqs\VBA
- 4) Double click vba71.msi
- 5) Double-click vba71-kb2783832-x64.msp
- 6) Double-click vba71_1033.msi*
Please make sure to follow this order. Also, install the language-specific VBA* by double-clicking the respective MSI file
- 7) Once done with the reinstall/repair of VBA, perform a repair of the SOLIDWORKS installation

*Install the following package for the desired language

vba71_1028.msi	Chinese
vba71_1031.msi	German
vba71_1033.msi	English
vba71_1036.msi	French
vba71_1040.msi	Italian
vba71_1041.msi	Japanese
vba71_1042.msi	Korean
vba71_1046.msi	Portuguese-Brazilian
vba71_2052.msi	Chinese-Simplified
vba71_3082.msi	Spanish

Please note that the default download location is C:\users\\Documents\SOLIDWORKS Download\.

Question: How can I contact the Student Edition support team?

Answer: You can contact the Student Edition Support Team by filling in the student support form using this link and providing the required information: https://www.solidworks.com/sw/education/All_product_support_req_form.html

When contacting the Student Edition Support Team, please collect and include the following information:

For installation and download issues:

1. Installation Logs:

Please provide the installation log files for further investigation of your issue.

To go to the installation log folder, you can type %appdata% at Start > RUN and press Enter; it will open the "Roaming" folder. In the "Roaming" folder go to the "SOLIDWORKS" folder. Under the "SOLIDWORKS" folder, you will find a folder named "Installation log." Zip this complete folder and provide it for investigation.

2. Screenshot of the error message

3. Serial numbers

For Crash and SOLIDWORKS functionality issues, or SOLIDWORKS files related issues:

1. SOLIDWORKS Rx:

Please capture your problem using the Rx utility and send the zip file back for further investigation.

(Default location of Rx is Start > All Programs > SOLIDWORKS Student Edition > SOLIDWORKS Tools > SOLIDWORKS RX)

Run Rx utility and go to "Problem Capture" and Click "Record Video." It will launch SOLIDWORKS and you can follow steps to show your issue. Click "Finish recording" after you have followed the steps to show the issue. Then Click "Package Files Now..." and save the zip file. Now under "Describe Problem" click "Continue" and enter the problem description and required details and click "OK" and exit the Rx utility. Send the zip file as an attachment for further investigation.