

Packages - Bug #3567

freecad doesn't work because of incompatible opencascade libs

2024-01-23 03:15 AM - Mampir

Status: fixed	% Done: 0%
Priority: bug	
Assignee: bill-auger	
Category:	
Description	
When you run freecad, the GUI starts, but it displays the following error:	
During initialization the error "libTKMesh.so.7: cannot open shared object file: No such file or directory" occurred in /home/mmp/.local/share/FreeCAD/Mod/kicadStepUpMod/./InitGui.py	
Practically nothing works, because of this error. The reason for the error is because the current opencascade package is not compatible with the current freecad package. The upstream Arch package has a patch included, which fixes this issue. I tried to build freecad-0.21.2 with the upstream patch and it worked for me:	
https://gitlab.archlinux.org/archlinux/packaging/packages/freecad/-/blob/main/PKGBUILD	
I also have to remind that there was another file called branding.xml that was added into the Parabola's PKGBUILD file, but it's not included into the Git repository. I've attached a patch with all the changes needed to build freecad-0.21.2	

History

#1 - 2024-01-25 01:29 AM - bill-auger

- Assignee set to bill-auger

- Status changed from unconfirmed to in progress

i pinned freecad to the version of opencascade - that should prevent breakage in the future

#2 - 2024-02-13 12:21 AM - bill-auger

- Status changed from in progress to fixed

Files

freecad.patch	2.45 KB	2024-01-23	Mampir
---------------	---------	------------	--------