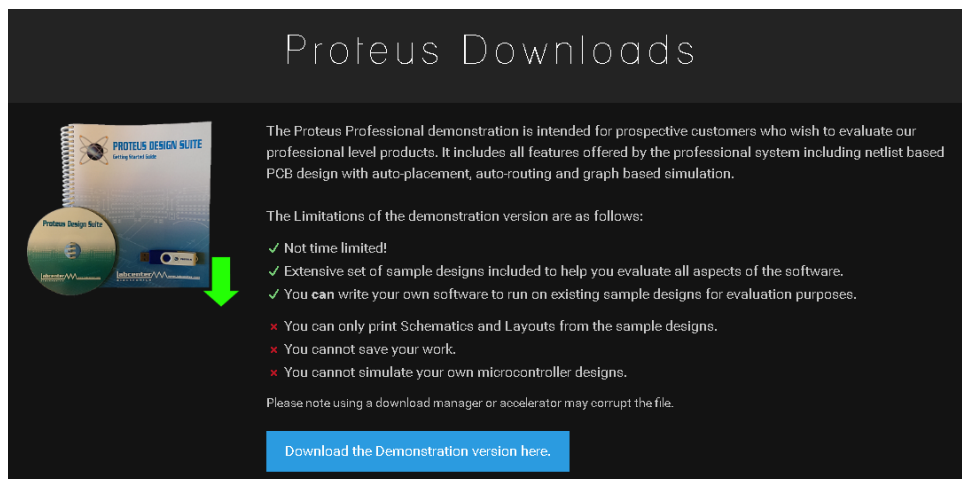




Tutorial para instalação do Proteus Design Suite 8

Link para download: <https://www.labcenter.com/downloads/>



Proteus Downloads

The Proteus Professional demonstration is intended for prospective customers who wish to evaluate our professional level products. It includes all features offered by the professional system including netlist based PCB design with auto-placement, auto-routing and graph based simulation.

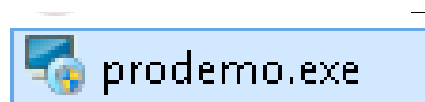
The Limitations of the demonstration version are as follows:

- ✓ Not time limited!
- ✓ Extensive set of sample designs included to help you evaluate all aspects of the software.
- ✓ You can write your own software to run on existing sample designs for evaluation purposes.
- ✗ You can only print Schematics and Layouts from the sample designs.
- ✗ You cannot save your work.
- ✗ You cannot simulate your own microcontroller designs.

Please note using a download manager or accelerator may corrupt the file.

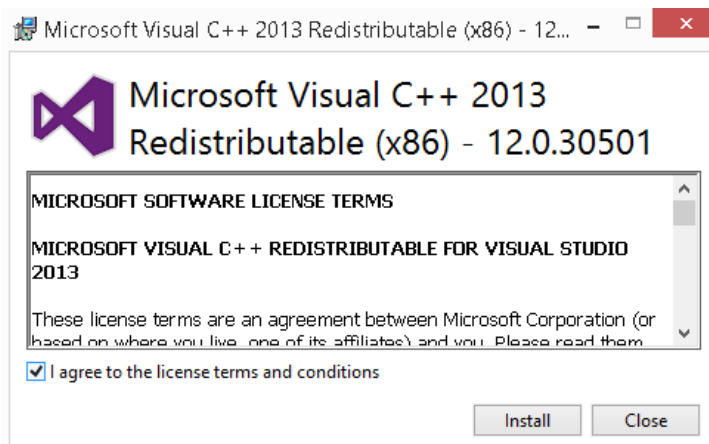
[Download the Demonstration version here.](#)

1. Clique em “Download the Demonstration version here” para iniciar o download.
2. Após o download, dê dois cliques no arquivo baixado:

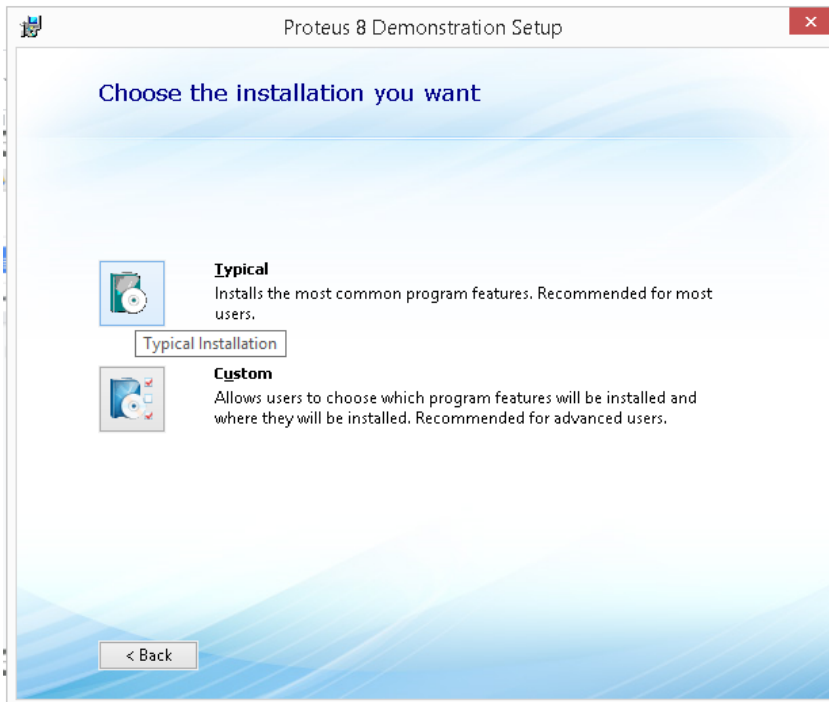


30 itens | 1 item selecionado 25,8 MB

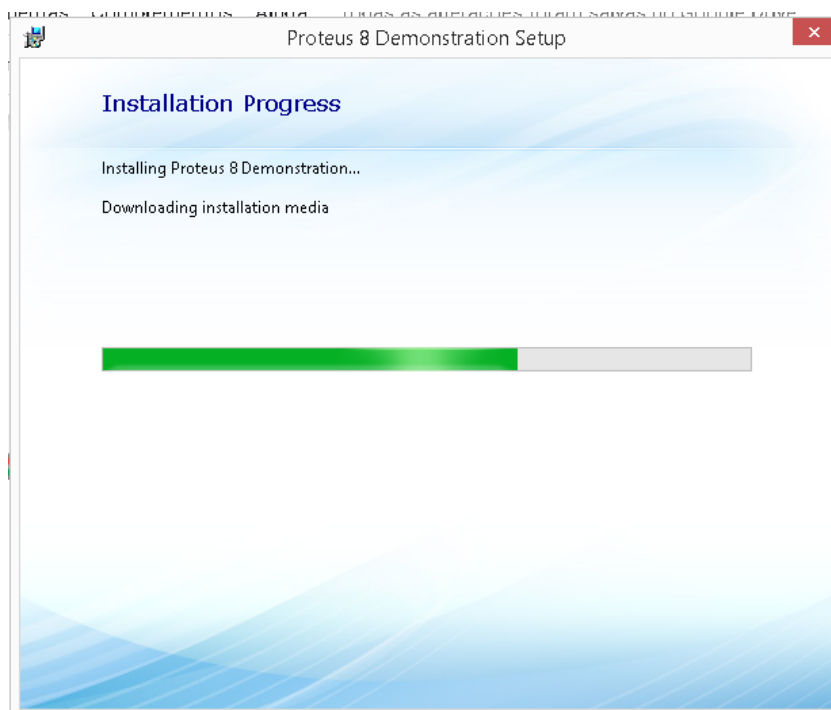
3. Uma tela de instalação do Visual Studio C++ irá aparecer. Aceite os termos de licença, marcando a caixa “I agree the license terms and conditions” e clique em “Install”.



4. Após a instalação da Microsoft Visual, escolha o tipo de instalação do Proteus. Escolha a opção “Typical”.



5. Aperte “Next”, para prosseguir.
6. A instalação do Proteus iniciará. Pode demorar um pouco para a conclusão da instalação.



7. Após alguns minutos: Instalação concluída.