
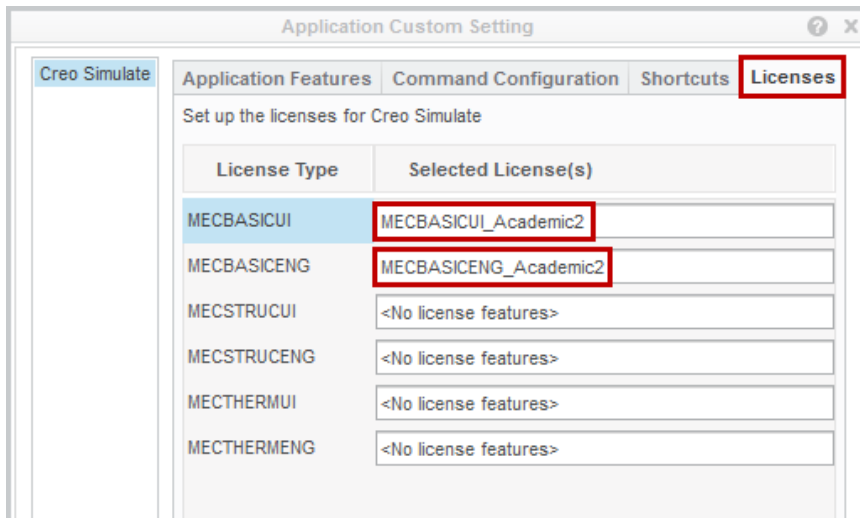


## Configuring Creo 2.0 Academic edition to run Simulate

1. Use Windows Explorer and browse to **C:\Program Files\PTC\Creo 2.0\Simulate\bin\**.
2. Run **reconfigure.exe**:
  - When using Windows 7, do this by right-clicking **reconfigure.exe** and selecting  **Run as administrator** from the pop-up menu.
  - If necessary, click **Yes** from the User Account Control dialog box.
3. From the PTC Installation Assistant dialog box:
  - Click **Next**.
  - Click **Next**.
  - Click **Customize**.
4. From the PTC Installation Assistant dialog box, open the **Licenses** tab:
  - Edit the Selected License(s) to be only the two shown below:



- Click **OK**.
- Click **Install**.
- Click **Finish**.

## You can now use Simulate from the Creo Parametric Applications tab

- If you have any questions, please open a call at <http://www.ptc.com/go/academic-support>.
- For additional resources, please visit <http://www.ptc.com/go/academicprogram>.