Download OpenFOAM

- You should now make sure that you know how to install OpenFOAM yourself.
- We will download the 32-bit version, but we will re-use the ThirdParty directory already in the system, so that we avoid some problems installing the parts that are not really part of OpenFOAM. Installation of the Third-Party products can be quite system-dependent, so you should try that in your own system (not part of this course). We will install in `/scratch` since your quota is limited. Then we will delete the installation immediately.
- Open `www.openfoam.org` and download the **32-bit Double Precision version** to `/scratch/$USER/OpenFOAM` according to the instructions (you need two files - exclude the ThirdParty files). I recommend that you for each download link: click on the link, click on cancel, right-click on `direct link`, click on `Properties`, copy the direct `Address` to a terminal window, where you type: `wget <Address>`. This will make sure that you get the complete file. You can also use this line in a script for automatic installation...
- Unpack the files according to the instructions at the download page.
- `rm *.gtgz` removes the packed files.

Install OpenFOAM

- Edit file `/scratch/$HOME/OpenFOAM/OpenFOAM-1.6/etc/bashrc`. Find where `foamInstall` is set, and edit it so that it points at your installation.
- Find in the same file where `WM_THIRD_PARTY_DIR` is set, and edit it so that it points at `/chalmers/sw/unsup/OpenFOAM/ThirdParty-1.6` (since we want to use the pre-installed Third-Party products)
- Edit your `~/.bashrc` file to source the `bashrc` file of your own installation using alias `OF16`.
- Open a new terminal window, from which you type:

  ```
  OF16
  which icoFoam
  ```

  This should give:
  `/scratch/$USER/OpenFOAM/OpenFOAM-1.6/applications/bin/linuxGccDPOpt/icoFoam`
- Now you are ready to go with your own installation of the pre-compiled distribution. Try to run the `icoFoam/cavity` tutorial. (paraFoam will not work, but we don't care about that at the moment)
- Delete all the files you have created in `/scratch`, and delete the `OF16` alias in your `~/.bashc` file.

Write a script to download and install OpenFOAM (1/2)

- If you can do all the steps using Linux commands, you will be able to make a script to automatically download and install OpenFOAM. You should do this as part of your assignment for the next occasion!!!
- Create a file named `downloadAndInstallOpenFOAM` (you should send this file to me later!)
- Add to the first line:
  `#!/bin/sh`
  (This specifies which shell to use in the script)
- Add all the command-lines you needed for the download and installation.
- We did some things by hand, so we have to learn how to do this on command-line. We edited a `bashrc` file. We can use `sed` to do this, and we can find examples of the use of `sed` in the `icoFoam/Allrun` script, by typing `man sed`, or searching the Internet. Substitute a string in a file to another string, by doing something similar to:

  ```
  sed -i s/'foamInstall=$HOME\/$WM_PROJECT'/\
  'foamInstall=\/scratch\/hani\/OpenFOAM'/g bashrc
  ```

  Note: Make sure that your use the exact path to your (`$USER`) installation!

Write a script to download and install OpenFOAM (2/2)

- You can see that we need to use \ before special character /. Here I have also used \ to continue the expression on a second terminal line, so write the expression exactly as above, on two lines, or put it on a single line and remove the \ at the end of the first line (make sure that there is no `space` left!).
- Instead of modifying the `~/.bashrc` file, **do the sourcing inside the script** by adding something similar to the following line:

```
. /scratch/$USER/OpenFOAM/OpenFOAM-1.6/etc/bashrc
```

- Now you are able to continue the script with any OpenFOAM commands you like, for example:

```
which icoFoam
run
cd cavity
blockMesh
icoFoam > log
```

(end the script with an empty line to make sure that all lines are run!)
- On command-line, type: `chmod +x downloadAndInstallOpenFOAM` to make it into an executable
- Run the script in a terminal window by typing: `./downloadAndInstallOpenFOAM`