

## **Start Turbomachinery CFD Step by Step**

See also: <http://www.cfdsupport.com/turbomachinery-cfd-workflow.html>

### **In Windows:**

1. Request the trial version: <http://www.cfdsupport.com/turbomachinery-cfd-demo.html>
2. Download and Install *OpenFOAM® for Windows* (including *Paraview* and *Gnuplot* options):  
<http://www.cfdsupport.com/download-openfoam-for-windows.html>
3. Download and Install *Turbomachinery CFD Demo* (note: software versions have to match; for example: *OpenFOAM for Windows 17.02* fits to *Turbomachinery CFD 17.02*)
4. Download a tutorial: <http://www.cfdsupport.com/download-cases.html>
5. Copy two demo license files (*License.key* and *License.dat*) to the current *OpenFOAM* version directory, typically: *C:\OpenFOAM\17.02\\**
6. Ready to run now
7. Extract your tutorial into work directory, for example: *C:\OpenFOAM\17.02\user-3.0.x\run\pump*
8. **When Using GUI:**
9. Open configuration file e.g. *pump.tcf* in *ParaView*

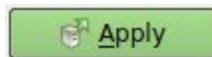
10. Click *Apply* button



11. In pipeline browser click *settings* and then apply *TCFD Manager* filter



12. Click *Apply* + *Write Case* + *Run All*



+ *Write case*

+ *Run all*

13. And all the process is done automatically: new case is written into default case name of *case0*, mesh is created, case is set-up, case is simulated, results are evaluated and report is written down.

14. The report can be updated anytime during simulation

*Update report*

15. When the simulation is finished the final results' report is written down. It is located inside the test case:

*./ofcase0/report-ofcase0/ofcase0.html*

16. Detailed visual postprocessing can be done using [Turbo Blade Post](#) filters (included).

### **8. When Using Terminal:**

9. Run [OpenFOAM® for Windows](#)

10. Navigate e.g. to the pump directory (*\$ mc* midnight commander is recommended, or go manually in terminal) and run all by one command from the *pump* directory:

```
$ CFDProcessor -setup pump.tcf -allrun
```

11. And all the process is done automatically: new case is written into default case name of *case0*, mesh is created, case is set-up, case is simulated, results are evaluated and report is written down. When the

simulation is finished the final results' report is written down immediately. It is located inside the test case:  
`./ofcase0/report-ofcase0/ofcase0.html`

12. Visual postprocessing can be done in [ParaView](#) and in [Turbo Blade Post](#) (included).

## In Linux:

1. Request the trial version: <http://www.cfdsupport.com/turbomachinery-cfd-demo.html>
2. Download `TurbomachineryCFD-17.02-demo-Linux.tar.gz`
3. In terminal extract into your favorite directory (e.g. `/home/michael/`):  
`$ tar xf TurbomachineryCFD-17.02-demo-Linux.tar.gz`
4. Download the tutorial: <http://www.cfdsupport.com/download-cases.html>
5. Copy two license files (`License.key` and `License.dat`) to the `TurbomachineryCFD-17.02-demo-Linux` directory
6. Ready to run now
7. In a new terminal source the system variables (with every new terminal, or add this to your `.bashrc` for permanent usage):  
`$ source OpenFOAM/OpenFOAM-3.0.x/etc/bashrc-release`
8. Extract your tutorial for example into your run directory:  
`/home/michael/OpenFOAM/michael-3.0.x-cfdsupport/run/pump`

### 9. When Using GUI:

10. Open configuration file e.g. `pump.tcf` in `ParaView`

11. Click `Apply` button



12. In pipeline browser click `settings` and then apply `TCFD Manager` filter



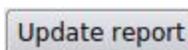
13. Click `Apply` + `Write Case` + `Run All`



14. And all the process is done automatically: new case is written into default case name of `case0`, mesh is created, case is set-up, case is simulated, results are evaluated and report is written down.

15. When the simulation is finished the final results' report is written down immediately. It is located inside the test case: `./ofcase0/report-ofcase0/ofcase0.html`

16. The report can be updated anytime during simulation



17. Detailed visual postprocessing can be done using [Turbo Blade Post](#) filters (included).

### 9. When Using Terminal:

10. In terminal navigate to the pump directory (`$ mc` - midnight commander is recommended, or go manually in terminal) and run all by one command:

```
$ CFDPprocessor -setup pump.tcf -allrun &
```

11. And all the process is done automatically: new case is written into default case name ofcase0, mesh is created, case is set-up, case is simulated, results are evaluated and report is written down. When the simulation is finished the final results' report is written down immediately. It is located inside the test case:

```
./ofcase0/report-ofcase0/ofcase0.html
```

12. Visual postprocessing can be done in [ParaView](#) and in [Turbo Blade Post](#) (included).

## General Notes & Recommendations:

1. **IN CASE OF ANY ISSUES, PLEASE SEND AN EMAIL WITH THE DESCRIPTION AND ATTACHED ZIPPED *logRun* FOLDER TO [INFO@CFDSUPPORT.COM](mailto:INFO@CFDSUPPORT.COM)**
2. See also: <http://www.cfdsupport.com/turbomachinery-cfd-workflow.html>
3. **Turbomachinery CFD input** is the surface geometry and physical settings being read from **Turbomachinery CFD Configuration File (\*.tcf)**
4. **CPU time estimation** - the total estimated time of the running simulation is written and updated every minute in file: *./logRun/timing*
5. **Log files** - are located in *./logRun/\**
6. **Number of parallel processors** - is set in configuration file within parameter *processors*
7. **Mesh size** - the most important mesh parameter of each component is *O\_backgroundMeshSize*, which is a basic mesh block (cell) to be refined. Three dimensions *x y z* in meters (*scaleFactor* parameter may change the dimension).
8. **CFD Processor** has several options. Custom case name. Writing the case. Writing the mesh. Run the simulation. Write the report. See all the options: *\$ CFDProcessor -help*
9. Any number of surface model components is allowed
10. Each component has to be **waterproof**
11. Ideally, each component surfaces should fit vertex-to-vertex. Not necessary, but safest.
12. For any CFD simulations it is not suitable to have "Trailing edge fixed on outlet" of the Impeller. It is recommended, when simulate, to extend the hub and shroud a little bit.
13. .STL surface model files are recommended in **ASCII format**
14. **compressibility**: pump and waterTurbine are always incompressible, compressor always compressible, fan any, turbine any
15. **rotatingPatches**: If *rotatingRegion* is false. RotatingPatches (if any) have *rotatingWallVelocity* (symmetric rotation – no MRF)
16. **Rotation direction** has signum minus for clockwise direction (right hand rule)
17. All the **interfaces** between two components **should overlap**, or at least to fit each other very well

## When using **CFturbo**: Recommended export procedure:

1. *Must do*: Make sure the interfaces fit
2. *Recommended*: Impeller - Main Dimensions - Setup - off Unshrouded
3. *Must do*: Impeller - CFD setup - Activate RSI (radial is safest, longer is better)
4. *Must do*: Impeller - Model Finishing
5. *Must do*: Project - Export - Interfaces - Turbomachinery CFD
6. *Must do*: Project - Export - check all Components & set Export destination & set Base file name (no special characters)
7. *Recommended*: Project - Export - Set parameters - Max. element length to minimum value allowed
8. *Recommended*: Project - Export - Granularity set to Very fine
9. *Must do*: Project - Export - Export data
10. *Recommended*: copy the *gnuplot* scripts folder from corresponding Turbomachinery CFD tutorial to your newly exported case
11. *Recommended*: set the number of parallel processors
12. *Recommended*: run the case