

HyperCFD
Fertinaz Yazılım Ltd.

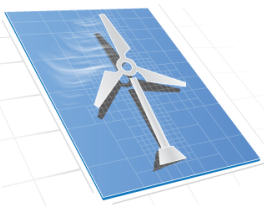
OpenFOAM Workshop Teknopark Istanbul

07 Conclusion
January 2017



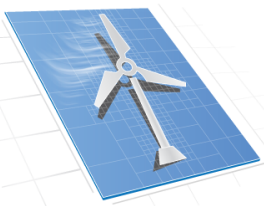
HyperCFD – Disclaimer

This offering is not approved or endorsed by OpenCFD Limited, the producer of the OpenFOAM software and owner of the OpenFOAM and OpenCFD trademarks.



HyperCFD – OpenFOAM Conclusion

- A good mesh is vital to produce good results
- And its necessary to have a clean and smooth geometry to generate a good mesh
- Therefore it all starts with having a good geometry
- Try to avoid sharp edges
- Keep your boundaries where things are happening
- Create your mesh carefully, go step by step if you're unsure
- Use `checkMesh` and `ParaView`
- Consider matrix reordering `renumberMesh`



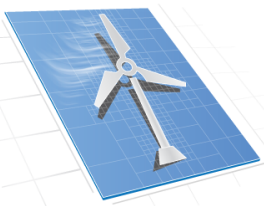
HyperCFD – OpenFOAM Conclusion

- To run parallel in computer clusters do not forget to source OpenFOAM by default
- Always use log files
- Use `grep`, `find`, `tail`, `sed`, `awk`, `top`, `less`, `more`, `vi`
- Find the most similar tutorial for your problem and change parameters carefully

```
cp -r $FOAM_TUTORIALS/multiphase/interDyMFoam/ras/DTCHull \  
$WM_PROJECT_USER_DIR/run/
```

```
cd $WM_PROJECT_USER_DIR/run/
```

```
mv DTCHull my_hull
```



HyperCFD – OpenFOAM Conclusion

- Use realistic initial conditions
- If you will run a transient case, use a converged steady state solution as an input
- Consider the CFL condition in transient cases
- Pay attention to the discretization schemes and solver settings
- Official OpenFOAM documentation is not huge but still has it provides lots of information
- But if you need more there lots of documents you can find online
- And there are consultants 😊
- Thank you

