

Using Template Files in PyFoam

Spoken Tutorial Project

<http://spoken-tutorial.org>

National Mission on Education through ICT

<http://sakshat.ac.in>

Script: Anmol Sahoo, Rahul Joshi

Narration: Rahul Joshi



Learning Objectives

- Understand the function of PyFoam Utilities



Learning Objectives

- Understand the function of PyFoam Utilities
- Create and use template files



Learning Objectives

- Understand the function of PyFoam Utilities
- Create and use template files
- Use `PyFoamFromTemplate.py` for solving the supersonic flow over wedge



Learning Objectives

- Understand the function of PyFoam Utilities
- Create and use template files
- Use PyFoamFromTemplate.py for solving the supersonic flow over wedge
- We can run this for different wedge angles using template files



System Requirement

- **Ubuntu Linux Operating system 14.04**



System Requirement

- **Ubuntu Linux Operating system 14.04**
- **OpenFOAM v2.3.0**



System Requirement

- **Ubuntu Linux Operating system 14.04**
- **OpenFOAM v2.3.0**
- **PyFoam 0.6.5**



Pre-requisite

- Basic knowledge of using Linux terminal



Pre-requisite

- **Basic knowledge of using Linux terminal**
- **Experience running and analyzing OpenFOAM cases**



Template Files

- Template files are used to generate OpenFOAM files like



Template Files

- Template files are used to generate OpenFOAM files like
- **blockMeshDict** or **controlDict**



Template Files

- Template files are used to generate OpenFOAM files like
- blockMeshDict or controlDict
- Template files can be programmed, hence we can procedurally generate data



Syntax of Template Files

A template file should be like an OpenFOAM file with the following additions:

- Any line beginning with \$\$ is a Python program line.



Syntax of Template Files

A template file should be like an OpenFOAM file with the following additions:

- Any line beginning with \$\$ is a Python program line.
- It will be executed by Python.



Syntax of Template Files

- Any variable can be substituted in the file by using the syntax
| *—varname—* |



Using a template file

The following steps are followed to use a template file:

- **Copy an existing file**



Using a template file

The following steps are followed to use a template file:

- Copy an existing file
- Create a template file



Using a template file

The following steps are followed to use a template file:

- Copy an existing file
- Create a template file
- Run `PyFoamFromTemplate.py`



Creating a Template File

- **Template file will be created for blockMeshDict**



Creating a Template File

- **Template file will be created for blockMeshDict**
- **We will use the supersonic flow over a wedge as an example case**



Creating a Template File

- Template file will be created for blockMeshDict
- We will use the supersonic flow over a wedge as an example case
- The case file is available in the rhoCentralFoam solver



Vertices to be changed

Based on the angle change the following lines

- (0.3048 0.081670913853 -0.005)
- (0.3048 0.081670913853 0.005)



Template syntax

- `$$ import math`
- `$$ thetaDegrees = 10.0`
- `$$ thetaRadians =
math.radians(thetaDegrees)`
- `$$ y_coord =
0.3048*math.tan(thetaRadians)`



Template syntax

Modify the vertices entry as

- (0.3048 |-y_coord-| -0.005)
- (0.3048 |-y_coord-| 0.005)



Running PyFoam command

- **pyFoamFromTemplate.py template-file=./blockMeshDict.template
output=./blockMeshDict
values-dictionary=templateFileConst
allow-exec-instead-of-assignment**



Running the case file

We can now run the case file by executing the OpenFOAM commands

- **blockMesh**



Running the case file

We can now run the case file by executing the OpenFOAM commands

- blockMesh
- rhoCentralFoam



Running the case file

We can now run the case file by executing the OpenFOAM commands

- blockMesh
- rhoCentralFoam
- Visualize the results using Paraview



Assignment

As an assignment , use the following wedge angles and run the template commands

- 0° , 5° , 20° , 25°



Summary

- In this tutorial, we learnt about **PyFoam Template Files**
- Create and use template files
- Using the **PyFoamFromTemplate.py** command



Forum to answer questions

- Do you have questions on THIS Spoken Tutorial?
- Choose the minute and second where you have the question.
- Explain your question briefly.
- Someone from the FOSSEE team will answer them. Please visit <http://forums.spoken-tutorial.org/>



Forum to answer questions

- Questions not related to the Spoken Tutorial?
- Do you have general / technical questions on the Software?
- Please visit the FOSSEE Forum <http://forums.fossee.in/>
- Choose the Software and post your question.



Textbook Companion Project

- The FOSSEE team coordinates coding of solved examples of popular books
- We give honorarium and certificate to those who do this

For more details, please visit this site:

<http://cfd.fossee.in/>



Acknowledgements

- Spoken Tutorial Project is a part of the Talk to a Teacher project
- It is supported by the National Mission on Education through ICT, MHRD, Government of India
- More information on this Mission is available at:

<http://spoken-tutorial.org/NMEICT-Intro>



THANK YOU!

For more information, visit our website
<http://fossee.in/>

