

SU2 Analysis Interface Module (AIM)

Modified by: Nitin Bhagat

Originally created by Ryan Durscher
AFRL/RQVC for FUN3D

August 1, 2016

Contents

1	Introduction	1
1.1	SU2 AIM Overview	1
1.1.1	Automatic generation of SU2 Mesh file	1
1.1.2	Automatic generation of SU2 Configuration file	1
2	Geometry Fidelity	2
3	User Inputs For SU2 Configuration File	2
4	Post Analysis AIM Outputs	3
5	Data Transfer	3
6	CFD Boundary Conditions	3
6.1	JSON String Dictionary	3
6.1.1	Wall Properties	4
6.1.2	Wall Properties	4
6.1.3	Stagnation Properties	4
6.1.4	Static Properties	4
6.1.5	Velocity Components	4
6.1.6	Massflow Properties	4
6.2	Single Value String	4

1 Introduction

1.1 SU2 AIM Overview

This module can be used to interface SU2 with geometry in CAPS system. The SU2 AIM has both c-version and a python-version for the interface. For SU2 capabilities and related documentation, please refer to <http://su2.stanford.edu/>. SU2 expects a volume mesh file and a corresponding configuration file to perform the analysis.

1.1.1 Automatic generation of SU2 Mesh file

The volume mesh file from SU2 AIM is written in native SU2 format ("filename.su2"). The description of the native SU2 mesh can be found SU2 website. For the automatic generation of mesh file, SU2 AIM depends on Mesh AIMS, for example, TetGen or AFLR4/3 AIM.

1.1.2 Automatic generation of SU2 Configuration file

The configuration file ("filename.cfg") from SU2 AIM is automatically created by using the flow features and boundary conditions that were set in the driver program as a user input. For the rest of the configuration variables, default set of values are provided for a general execution. If desired, a user has freedom to manually (a) change these variables based on personal preference, or (b) override the configuration file with unique configuration variables.

Related pages:

- [CFD Boundary Conditions](#)
- geomFidelityFUN3D
- [Geometry Fidelity](#)
- [User Inputs For SU2 Configuration File](#)
- [Post Analysis AIM Outputs](#)
- sharableDataSU2

2 Geometry Fidelity

The geometric fidelity for the SU2 AIM requires that the body(ies) be either a face body(ies), solid body(ies) or non-and manifold sheet body(ies). For 2D simulations the attribute capsFidelity should be set to CFD with the geometric fidelity being either a planar SHEETBODY or FACEBODY, while for 3D simulations be set to CFD but the both SOLIDBODY(ies) or SHEETBODY(ies) are acceptable.

3 User Inputs For SU2 Configuration File

For the description of the configuration variables, associated values, and available options refer to the template configuration file that is distributed with SU2. Note: The configuration file is dependent on the version of SU2 used. This configuration file that will be auto generated is compatible with SU2 4.1.1.

Following variables are automatically set based on the input from driver program:

Keyword	Value
MESH_FILENAME	[Proj_Name].su2
MESH_FORMAT	ASCII

Following is a list of configuration variables that can be set by the driver program that will be used in creating SU2 configuration file.

Keyword	Default
Boundary_Condition	Refer CFD Boundary Conditions
Overwrite_CFG	false

Free stream conditions:

Keyword	Default
Mach	0.8
Alpha	0.0
Beta	0.0

Numerical Scheme:

Keyword	Default
Eqn	COMPRESSIBLE
Num_Iter	999
CFL_NUMBER	10.0
MGLEVEL	0
RESIDUAL_REDUCTION	6

Dimensionalization:

Keyword	Default
SYSTEM_MEASUREMENTS	SI
FREESTREAM_PRESSURE	101325.0 N/m ²
FREESTREAM_TEMPERATURE	288.15K
FREESTREAM_DENSITY	1.2886 Kg/m ³
FREESTREAM_VELOCITY	1.0 m/s
FREESTREAM_VISCOSITY	1.853E-5 N s/m ²
REF_DIMENSIONALIZATION	DIMENSIONAL
REF_ORIGIN_MOMENT_X	0.25 m
REF_ORIGIN_MOMENT_Y	0.00 m
REF_ORIGIN_MOMENT_Z	0.00 m
REF_LENGTH_MOMENT	1.00 m
REF_AREA	0, compute automagically

Output

Keyword	Default
OUTPUT_FORMAT	PARAVIEW

- **Two_Dimensional = False**
Run SU2 in 2D mode.

4 Post Analysis AIM Outputs

After successful completion, SU2 writes results in various files. The data from these files can be directly viewed, visualized, and or used for further postprocessing.

One of the files is ("forces_breakdown.dat") which summarizes convergence including flow properties, numerical paramters, and resulting force and moment values. As an AIM output, this file is parsed for force and moment coefficients, and printed as closing remarks.

5 Data Transfer

Data transfer blah, blah, blah....

6 CFD Boundary Conditions

Structure for the boundary condition tuple = ("CAPS Group Name", "Value"). "CAPS Group Name" defines the capsGroup on which the boundary condition should be applied. The "Value" can either be a JSON String dictionary (see Section [JSON String Dictionary](#)) or a single string keyword string (see Section [Single Value String](#))

6.1 JSON String Dictionary

If "Value" is a JSON string dictionary (eg. "Value" = {"bcType": "Viscous", "wallTemperature": 1}) the following keywords (= default values) may be used:

- **bcType = "Inviscid"**

Boundary condition type. Options: Inviscid, Viscous, Farfield, Extrapolate, Freestream, BackPressure, Symmetry, SubsonicInflow, SubsonicOutflow, MassflowIn, MassflowOut, FixedInflow, FixedOutflow.

6.1.1 Wall Properties

Something else

6.1.2 Wall Properties

- **wallHeatFlux = 0.0**

Heat flux on viscous surfaces.

6.1.3 Stagnation Properties

6.1.4 Static Properties

6.1.5 Velocity Components

6.1.6 Massflow Properties

6.2 Single Value String

If "Value" is a single string the following options maybe used:

- "Inviscid" (default)
- "Viscous"
- "Farfield"
- "Extrapolate"
- "Freestream"
- "SymmetryX"
- "SymmetryY"
- "SymmetryZ"