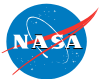


Session 8

Turbulent Flow Simulations

Jan-Renee Carlson



<http://fun3d.larc.nasa.gov>

FUN3D Training Workshop
April 27-29, 2010



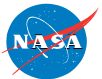
Session Overview

- Turbulent flow simulations
 - Available turbulence models
 - Namelist parameters
 - Typical simulations
 - Default values in namelists are **green**
 - Optional namelist arguments are in **blue**



Turbulent Flow Simulations

```
&governing_equations  
  eqn_type      = "cal_per_compress"  
  viscous_terms = "inviscid"  
  viscous_terms = "laminar"  
  viscous_terms = "turbulent"  
/
```



Turbulent Flow Simulations

```
&turbulent_diffusion_models
  turb_model          ="sa" (Spalart-Allmaras)
  turb_model          ="menter-sst" (k- $\omega$ -SST)
  turb_model          ="des" (Detached Eddy simulation )
  turb_model          ="hrles" (Hybrid RANS-LES)
  re_stress_model     ="linear"
  turb_compress_model ="off"
  prandtlnumber_turbulent = 0.9
```

/



Turbulent Flow Simulations

S-A

```
&turbulent_diffusion_models
  turb_model          ="sa"
  turb_compress_model ="none"
  turb_compress_model ="ssz" (AIAA-95-0863, Shur et al.)
/
&spalart
  turbinf = 1.3 (default makes  $v_{t,\infty} = 0.009$ )
/
```



Turbulent Flow Simulations

Menter-SST

```
&turbulent_diffusion_models
  turb_model          ="menter-sst"
  turb_compress_model ="none"
  turb_compress_model ="wilcox" (Wilcox compressibility)
/
&kw_sst
  strain_production =.false. (Default uses vorticity)
  strain_production =.true. (strain rate tensor)
  k_inf            = 9.0e-9
  w_inf            = 1.0e-6
/
```



Turbulent Flow Simulations

```
&turbulent_diffusion_models  
  turb_model          ="des"  
  turb_model          ="hrles"  
/
```

- Defer to after-session discussion as this topic is too extensive for the time allocated.



Typical Usages

- RANS wall bounded flow simulation using Spalart-Allmaras (part 1)

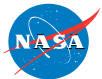
```
&version_number
  input_version          = 2.0
/
&project
  project_rootname      = "bump_3levelsdn_177x81"
/
&reference_physical_properties
  mach_number           = 0.2
  reynolds_number       = 3000000.0
  temperature           = 540.0
  temperature_units     = "Rankine"
/
&turbulent_diffusion_models
  turb_model            = "sa"
/
&spalart
  turbinf               = 3.0
/
&nonlinear_solver_parameters
  schedule_iteration    = 1      250
  schedule_cfl          = 10.    250.
  schedule_cfl_turb     = 10.    250.
/
&code_run_control
  steps                 = 20000
  restart_write_freq    = 1000
  restart_read          = "off"
/
```



Typical Usages

- RANS wall bounded flow simulation using Spalart-Allmaras (part 2)

```
&boundary_conditions
  total_pressure_ratio(3)    = 1.02828
  total_temperature_ratio(3) = 1.008
  static_pressure_ratio(4)  = 1.0
/
&boundary_output_variables
  mu_t = .true.
/
```



Typical Usages

- RANS shear/Jet flow simulation using k- ω -SST (part 1)

```
&raw_grid
  grid_format = "vgrid"
  data_format = "unformatted"
/
&project
  project_rootname = "arn2"
/
&inviscid_flux_method
  flux_limiter      = "none"
  flux_construction = "roe"
/
&turbulent_diffusion_models
  turb_model      = "menter-sst"
/
&code_run_control
  steps           = 500
  restart_write_freq = 250
  restart_read    = "on"
/
&reference_physical_properties
  mach_number     = 0.50
  reynolds_number = 0.5e+6
  temperature     = 300.0
/
```



Typical Usages

- RANS shear/Jet flow simulation using k- ω -SST (part 2)

```
&boundary_conditions
  total_pressure_ratio(1) = 1.197
  total_temperature_ratio(1) = 0.950
  static_pressure_ratio(4) = 1.0
/
&nozzle_parameters
  inflow_pt_ramp=100,
/
&sampling_parameters
  number_of_geometries = 1
  type_of_geometry(1) = 'circle'
  circle_center(1,:) = 0.,0.,0.
  circle_normal(1,:) = 1.,0.,0.
  circle_radius(1) = 5.
/
&sampling_output_variables
  mach = .true.
  turb1 = .true.
  turb2 = .true.
  mu_t = .true.
/
```



Typical Usages

- RANS shear/Jet flow simulation using k- ω -SST (part 3)

```
&boundary_output_variables
  number_of_boundaries = 3
  boundary_list        = '1, 7, 8'
  mach                 = .true.
  turb1                = .true.
  turb2                = .true.
  mu_t                 = .true.
  yplus                = .true.
/
&nonlinear_solver_parameters
  schedule_iteration = 1 500
  schedule_cfl       = 1.0 50.0
  schedule_cflturb   = 1.0 50.0
/
&version_number
  input_version = 2.2
/
```

- Execute statement

```
mpirun nodet_mpi \
  --animation_freq 100 \
  --sampling_freq 100 \
  --alternate_freestream 0.05 \
  > fun3d_output
```



Session 8 (cont.)

Boundary Conditions

(WE ARE THE PROBLEM!)

Jan-Renee Carlson

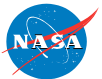


(WE'RE PART OF THE SOLUTION TOO!)



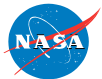
Session Overview

- Boundary conditions
 - Available boundaries (slightly abridged)
 - Namelist parameters (optional/needed) (slightly abridged)
 - A typical(?) simulation



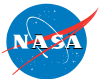
Boundary Conditions

	Grid	FUN3D	Type
<code>tangency</code>	5	3000	slip wall
<code>viscous_solid</code>	4	4000	no-slip wall
<code>symmetry_x,y,z</code>		6661, 6662, 6663	x-,y-,z-symmetry plane
<code>farfield_riem</code>	3	5000	Riemann
<code>farfield_roe</code>		5050	farfield
<code>back_pressure</code>		5051	outflow
<code>subsonic_outflow_p0</code>		7012	outflow
<code>subsonic_outflow_mach</code>		5052	outflow
<code>massflow_out</code>		7031	outflow
<code>massflow_in</code>		7036	inflow



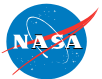
Boundary Conditions

	Grid	FUN3D	Type
<code>subsonic_inflow_pt</code>		7011	inflow
<code>subsonic_inflow_vel</code>		7010	inflow
<code>extrapolate</code>		5026	outflow
<code>fixed_inflow</code>		7100	inflow
<code>fixed_outflow</code>		7105	outflow



The Usual Suspects

- Farfield (typ. inflow) – **farfield_riem** (5000)
- Farfield (typ. outer boundary) – **farfield_roe** (5050)
- Wing/Body/Tail/Flate plate – **viscous_solid** (4000)
- Subsonic plenum / wind tunnel inflow – **subsonic_inflow_pt** (7011)
- Channel / wind tunnel outflow (allows supersonic flow)– **back_pressure** (5051)
- Supersonic inflow (e.g. nozzle exit face) – **fixed_inflow** (7100)
- Supersonic outflow – **extrapolate** (5026)
- Subsonic inlet (restricted to subsonic flow)– **subsonic_outflow_p0** (7012)



Sample Problem

Static test of an Acoustic Research Nozzle (ARN)

- Namelist settings
- Execution
- Output

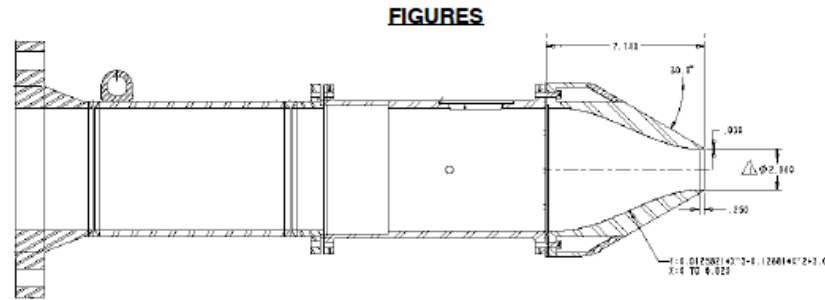
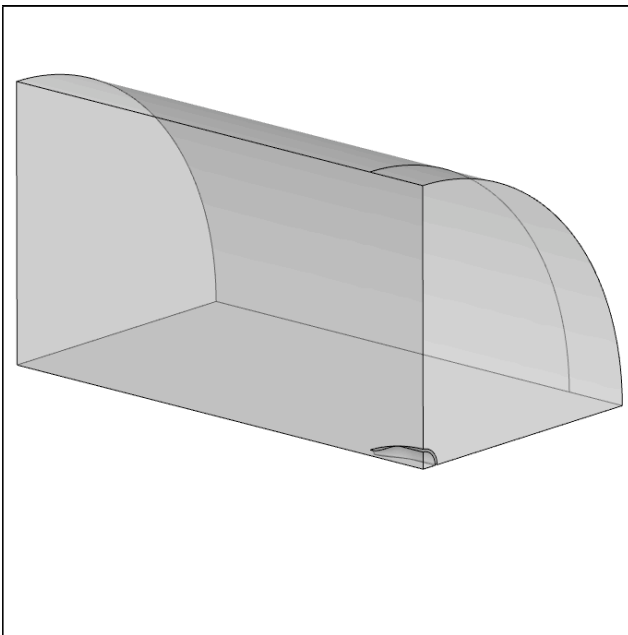


Figure 1 NASA Acoustic Reference Nozzle system, with ARN2 (51mm diameter) nozzle.

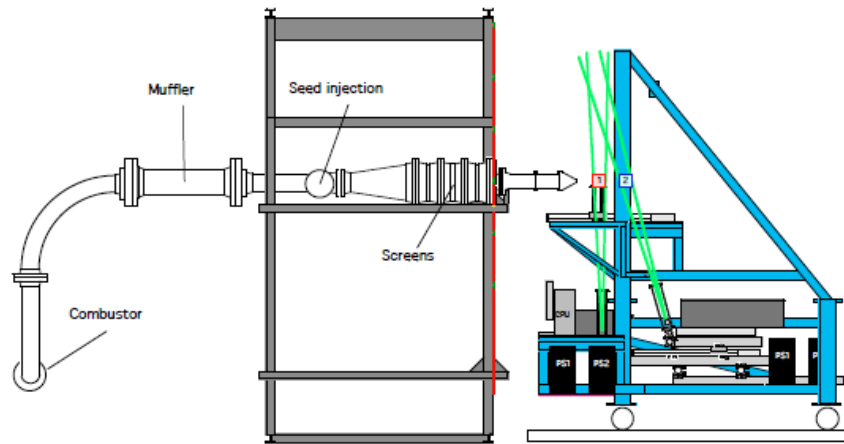


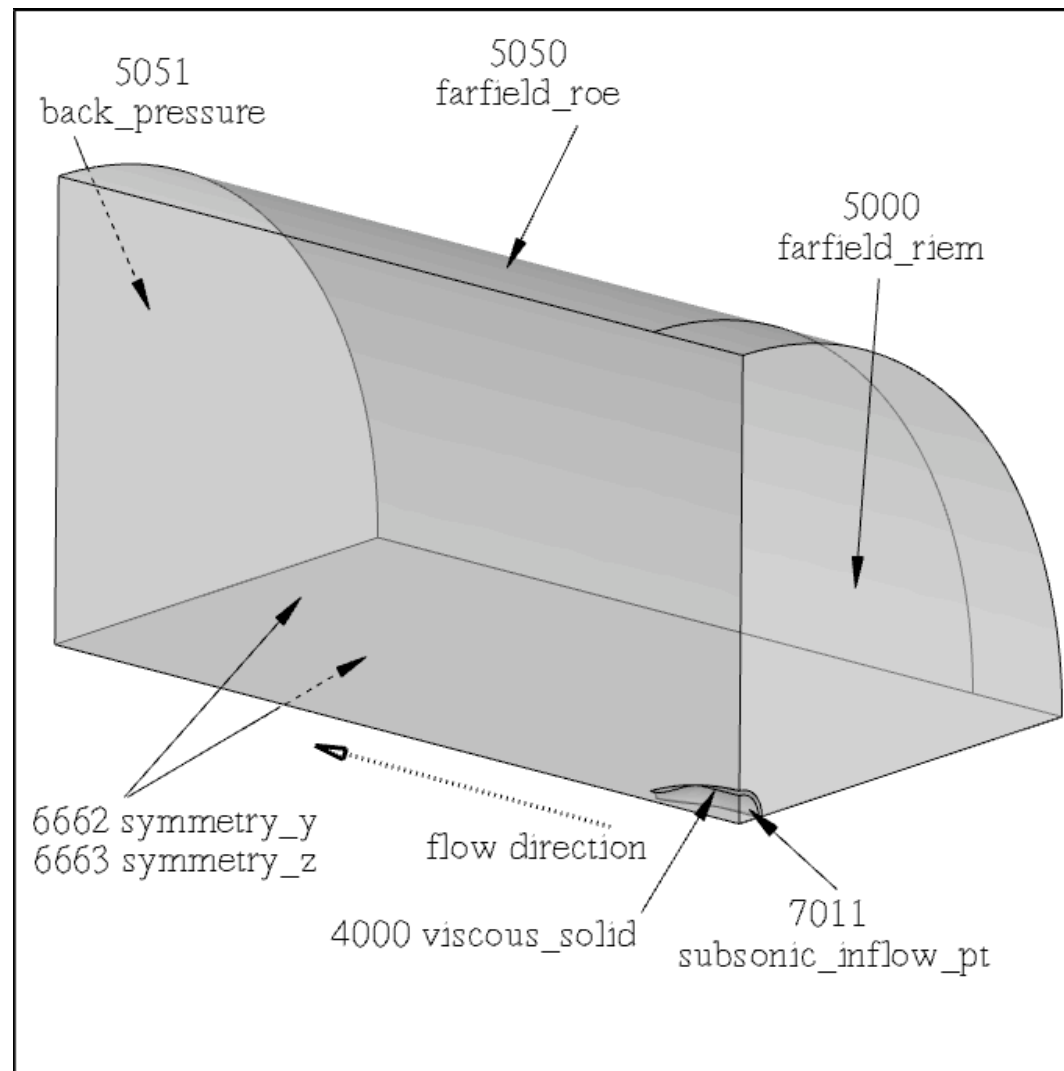
Figure 2 SHJAR with Dual PIV setup.

AIAA-2003-3130, Bridges

Sample Problem

Acoustic Research Nozzle

- Boundaries



Acoustic Research Nozzle

#Wed Jan 27 13:28:40 2010

#bc.map

Patch #	BC	Family	#surf	surfIDs	Family
1	7011	5	0	0	nozzleinflow
2	5050	3	0	0	farfield
3	5050	3	0	0	farfield
4	5051	3	0	0	outflow
5	4000	4	1	20	boattail
6	4000	4	1	19	nozzle
7	6663	1	0	0	z-symmetry
8	6662	1	0	0	y-symmetry
9	5000	3	0	0	inflow

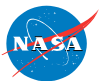
&boundary_conditions

total_pressure_ratio(1) = 1.197

total_temperature_ratio(1) = 0.950

static_pressure_ratio(4) = 1.0

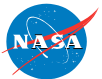
/



Acoustic Research Nozzle

- fun3d.nml

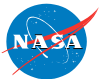
```
&raw_grid
  grid_format = "vgrid"
  data_format = "unformatted"
/
&version_number
  input_version = 2.0
/
&project
  project_rootname = "arn2"
/
&governing_equations
  viscous_terms = "turbulent"
/
&inviscid_flux_method
  flux_limiter = "none"
  first_order_iterations = 1000
  flux_construction = "roe"
/
&turbulent_diffusion_models
  turb_model = "menter-sst"
/
&code_run_control
  steps = 5000
  restart_write_freq = 250
  restart_read = "off"
/
```



Acoustic Research Nozzle

- fun3d.nml (cont.)

```
&sampling_parameters
  number_of_geometries = 1
  type_of_geometry(1)  = 'circle'
  circle_center(1,:)   = 0.,0.,0.
  circle_normal(1,:)   = 1.,0.,0.
  circle_radius(1)     = 5.
/
&sampling_output_variables
  mach   = .true.
  turb1  = .true.
  turb2  = .true.
  mu_t   = .true.
/
&boundary_output_variables
  number_of_boundaries=1
  boundary_list="1"
  mach   = .true.
  turb1  = .true.
  turb2  = .true.
  mu_t   = .true.
/
```



Acoustic Research Nozzle

- fun3d.nml (cont.)

```
&reference_physical_properties
  temperature_units = "Kelvin"
  mach_number      = 0.50
  reynolds_number  = 0.5e+6
  temperature      = 300.0
```

(Here Mach number and Reynolds number serve as the viscous scaling for the simulation since the jet is exhausting in to quiescent flow.)

```
/
&boundary_conditions
  total_pressure_ratio(1) = 1.197
  total_temperature_ratio(1) = 0.950
  static_pressure_ratio(4) = 1.0
```

```
/
&nozzle_parameters
  inflow_pt_ramp=100,
```

```
/
&nonlinear_solver_parameters
  time_accuracy      = "steady"
  schedule_number    = 2
  schedule_iteration = 1 500
  schedule_cfl       = 1.0 50.0
  schedule_cflturb   = 1.0 50.0
```



Acoustic Research Nozzle

- Execution

```
~/mpirun nodet_mpi \  
  --animation_freq 100 \  
  --sampling_freq 100 \  
  --alternate_freestream 0.05 \  
> fun3d_output
```

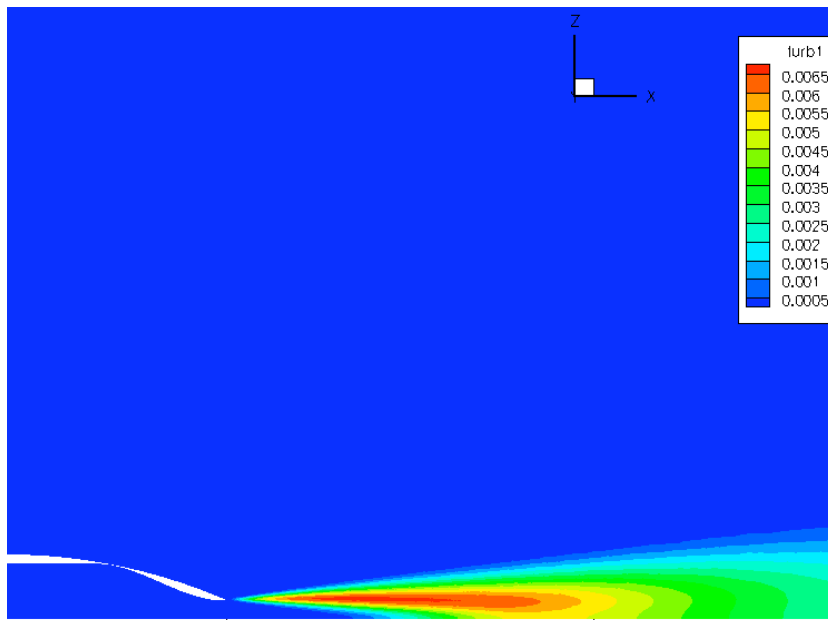
- Files output

- history file: `arn2_hist.tec`
- sampled data: `arn2_tec_sampling_geom1_timestepxxxx.dat`
- flowfield data: `arn2_tec_boundary_timestepxxxx.dat`

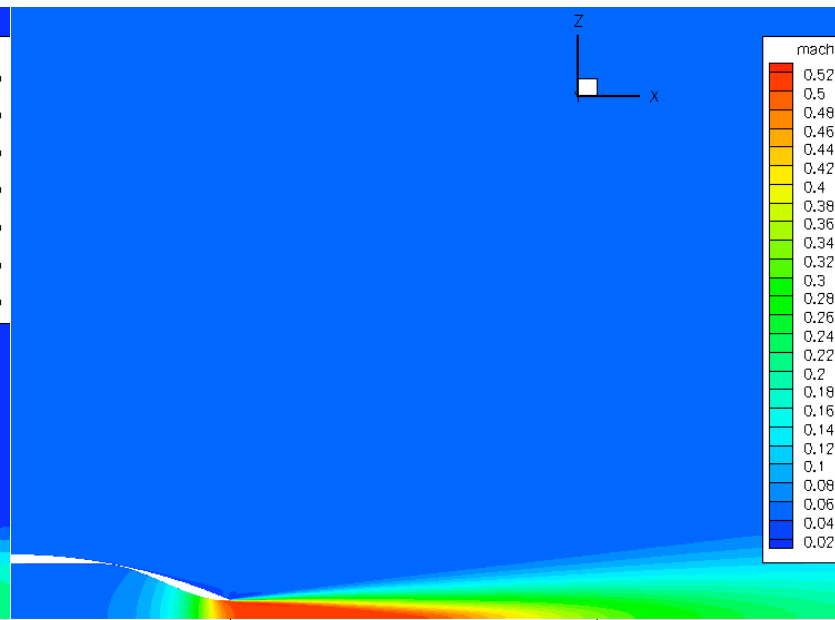


Acoustic Research Nozzle

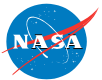
Turbulent kinetic energy



Mach number



Appendix: Boundary condition list



Boundary Conditions

`&boundary_conditions` namelist

Boundary	Input	Namelist parameter(s)
<code>tangency</code>	none	
<code>viscous_solid</code>	temperature (optional)	<code>wall_temperature(ib)</code> , <code>wall_temp_flag(ib)</code>
<code>symmetry_x,y,z</code>	none	
<code>farfield_riem,</code> <code>farfield_roe</code>	none	
<code>extrapolate</code>	none	



Boundary Conditions

`&boundary_conditions` namelist

Boundary	Input	Namelist parameter(s)
<code>back_pressure</code>	static pressure (Mach > 0)	<code>static_pressure_ratio(ib)</code>
<code>subsonic_outflow_p0</code>	static pressure (Mach < 1)	<code>static_pressure_ratio(ib)</code>
<code>subsonic_outflow_mach</code>	Mach # (0 < Mach < 1)	<code>mach_bc(ib)</code>



Boundary Conditions

`&boundary_conditions` namelist

Boundary	Input	Namelist parameter(s)
<code>massflow_out</code>	<code>massflow</code>	<code>massflow(ib)</code>
<code>massflow_in</code>	mass flow, total temperature	<code>massflow(ib)</code> , <code>total_temperature_ratio(ib)</code>



Boundary Conditions

`&boundary_conditions` namelist

Boundary	Input	Namelist parameter(s)
<code>subsonic_inflow_pt</code>	total pressure, total temperature, flow angle	<code>total_pressure_ratio(ib),</code> <code>total_temperature_ratio(ib),</code> <code>subsonic_inflow_velocity(ib)</code> <code>= "normal"</code>
<code>subsonic_inflow_vel</code>	density, velocity	<code>q_set(ib,1), q_set(ib,2:4)</code>



Boundary Conditions

`&boundary_conditions` namelist

Boundary	Input	Namelist parameter(s)
<code>fixed_inflow</code>	density, velocity & pressure ($M > 1$)	<code>q_set(ib,1:5)</code>
<code>fixed_outflow</code>	density, velocity & pressure (?)	<code>q_set(ib,1:5)</code>

