Wind-US Conjugate Heat Transfer Users Documentation

Introduction

A set of computational tools for coupling Wind-US with solid conduction and convection heat transfer has been developed. A modular approach is used in order that the tools can be used in a variety of combinations or in stand-alone applications. The key element to this approach is a library of Surface Data Interchange (SDI) routines callable by any of the computational modules. A loosely-coupled approach is used whereby the routines pass heat-transfer and temperature data, which become updated boundary conditions for the modules. The Surface Data Interchange (SDI) library is linked into each module to facilitate communication between the modules. The capability has been demonstrated on, and results compared with, data from two experiments. Examples for each of these cases are described below. The computational modules used are the Wind-US CFD code and the HTX solid heat conduction module. An interpolation tool is provided for surface grids that are not point matched.

More details of the model and comparisons with data can be found in the paper by Perrell, et al.

Installation

Download the CHT models from IVM in the “Wind Process” and extract in the $WIND_DEV directory. This will create a directory “models-dev”. Add the library link command (-lsdi –lsqlite3 -lsl) to the EXTRALIBS line in the appropriate Makefile.include.* file and remove “sdi” from the dummy library line in the source/Makefile.userfile.

A sample cshell login script, models.login, is provided in models-dev/bin. It should be modified as appropriate to set the $MODELSROOT environment variable to the desired installation directory. This script can then be sourced from the user’s .cshrc to set up the environment upon login.

To compile all of the codes and libraries in the models-dev directory type:

```
make all_models
```

or these can be made individually by specifying libsdi, htx, or sdi_interp_surface

Then make the Wind-US code as usual.

To install, source models.login first and type:

```
make install_models
```

The executables will be copied to the $MODESLROOT/$SYSTEM/$SYSTEM_CPU/bin directory. Documentation and scripts will be copied to the $MODELSROOT/etc/"tool"/doc and $MODELSROOT/etc/"tool"/scripts, respectively, for each tool, e.g. htx.

Setup and Execution

Environment Variables and Paths

The $MODELSROOT environment variable should be set to provide a path to documentation and scripts and the login script $MODELSROOT/bin/models.login should be sourced to set the path correctly.

Wind-US Keywords

The CHT capability is currently only available for structured grids using the TTSPEC keyword block, e.g.

```
TTSPEC
   Type Temperature
   Mode CHT
   Update Interval 10
   Zone 1
   Surface JMAX
ENDTTSPEC
```

This is a loosely-coupled implementation, so an interval should be specified between executions of the solid heat conduction code to minimize the possibility of unconstrained temporal oscillations at the surface. Specifying the CHT mode activates the data transfer capability and a Perl script in your local directory called "run_cht.pl" is executed at each coupling step.

CHT Script

A Perl script called "run_cht.pl" is required and should be located in the users run directory. Currently this script is NOT copied to a remote directory, so you must run with the –runinplace option. Example Perl scripts can be found in a directory specified by $MODELSROOT/etc/htx/scripts. Each of these is documented in-line. The typical execution flow is to interpolate the heat flux distribution from the CFD surface solution to the corresponding Heat Conduction surface, run the heat conduction code, and interpolate the resulting temperature distribution back to the CFD surface. The example script is:

- run_cht_htx.pl   Run the HTX code for a single surface

Communication between the modules is controlled by the interface surface name. The Wind-US surface name is hardwired to "wind_cht" and contains all of the surfaces that have been specified in the TTSPEC block as CHT. The surface name for the heat conduction code may be set to any name as specified in the input file. Be sure that the surface names in the heat conduction input are consistent with the names in the Perl script.

HTX Heat Conduction Code

The HTX solid heat conduction code solves the unsteady 3D heat conduction equations using a structured finite-volume formulation. In this formulation of the energy equation, no assumptions have been made regarding the material properties. In the cases considered here the thermal conductivity, density and specific heat are held as constants, but temperature dependence could easily be implemented.

The solid grid is a structured grid saved in the CGNS format (see http://cgns.sourceforge.net/) along with the solution. Many grid generation tools can generate CGNS files directly. You may also convert PLOT3D files to CGNS files using the CGNS utility 'adfviewer' available from the web site above.

Temperature and heat-transfer data along solid/fluid boundaries are exchanged between modules via SDI library routines. The user can specify any of the structured block sides for surface data interchange. The surface names and the Perl script are used to ensure proper communication.

See the documentation on HTX for more details.

Interpolation Code

The "sdi_interp_surface" code is provided to facilitate communication between fluid/solid interface surfaces that are not point matched. If the surfaces are point matched, then the surface name in the Heat Conduction code can be the same as that in Wind-US, i.e. "wind_cht" and the interpolation code is not required. Otherwise, the interpolation code is called with the arguments specifying the donor surface and receptor surface, e.g.

```
Sdi_interp_surface wind_cht htx_1
```

interpolates the solution from the Wind-US surface to the Heat Conduction surface called htx_1.

Execution

If the heat conduction code and sdi_interp_surface are in your path and Wind-US has been compiled with the Conjugate Heat Transfer option, then Wind-US is executed as usual. Be sure to specify run-in-place:

```
wind –runinplace
```

All of the communication occurs on the Master process, so the Conjugate Heat Transfer capability will work in parallel with either PVM or MPI.

Example Cases

Two example cases are provided. See the paper by Perrell, et al. for additional details.

JPL Nozzle

Data for the JPL water-cooled nozzle is available from Back, et al. While the nozzle is water cooled, there is no data available on the flow rates within the water jacket, so the outer surface temperature distribution is specified from the experimental data. The coolant flow example is only loosely based on the experimental information available. The fluid conditions and properties are as follows:

- P_0 = 517107. N/m2 = 75.0 lbf/in^2
- T_0 = 843.0˚ K = 1518.0˚ R
- M_{exit} = 2.2
- γ = 1.35

The solid properties are as follows:

- ρ = 7750. kg/m^3
- k = 36.67 W/m/˚ K
- C = 460.0 J/kg/˚ K

The outer temperature of the shell is specified.

The test case is in the directory "jpl_slice". A nearly converged solution is provided for the case. To verify operation of the CHT capability, the case has been run from this solution for 500 steps and the results saved in files with the extention ".result". To rerun the case first clean up and initialize the solution by typing:

```
./clean.csh
```

This will delete any leftover files, copy the initial CFL file, and generate any initial solid data that is required. For specified wall temperature distributions, a program called SDI_add_Touter is compiled and executed which generates the initial SDI files for the outer wall temperature from the distribution in the text file "tw_outer.dat". See the scripts and Fortran code for more details.

Now run the code for the appropriate case, jpl_slice.

A script "plot_temp.csh" is provided to generate surface data and plot against the experimental data. A set of Fieldview scripts called "fv_both" is provided to visualization the solid and fluid solutions. See the README file for more information.

HEAT-H1 Nozzle

The HEAT-H1 facility nozzle has been studied previously by Shope using a coupled Euler/Boundary Layer/Heat Conduction method with a water cooling model. This case was chosen for comparison to Shope's model, CCCP, and the commercial CFD code Fluent. The flow conditions and properties for this case (corresponding to Shope's Case I) are as follows:

- P_0 = 126 atm.
- T_0 = 5000.0˚ K = 9000.0˚ R
- M_{exit} = 1.8
- γ = 1.24

The solid properties are as follows:

- ρ = 8885 kg/m^3
- k = 390 W/m/K

- C = 385. J/kg/K

The coolant flow is given by:

- m = 5.234 kg/s
- p = 68 atm
- T = 309 K

The channel width is 0.075 inches. A coolant model is not available yet for HTX, so a specified outer wall temperature is provided.

The test case for the HEAT-H1 simulation is in the directory, heat_h1_slice_touter.

As before, a nearly converged solution is provided. To verify operation of the CHT capability, the case has been run from this solution for 500 steps and the results saved in files with the extention ".result". To rerun the cases first initialize the solution by typing:

```
./clean.csh
```

This will delete any leftover files, copy the initial CFL file, and generate the outer wall data.

Now run the code for the appropriate case, heat_h1_slice.

A script "plot_temp.csh" is provided to generate surface data and plot against the experimental data. A set of Fieldview scripts called "fv_both" is provided to visualization the solid and fluid solutions. See the README file in each directory for more information.

References

Back, Massier, and Gier, "Convective Heat Transfer in a Convergent-Divergent Nozzle," A. NASA TR 32-415.

Engblom, W. A., Fletcher, B., and Georgiadis, N.J., "Validation of Conjugate Heat-Transfer Capability for Water-Cooled High-Speed Flows," AIAA-2007-4392, 39th AIAA Thermodynamics Conference, 25-28 June 2007, Miami, FL.

Perrell, E. R., Power, G. D., and Robinson, C. , "A Modular Conjugate Heat Transfer Capability for the Wind-US CFD Code," AIAA-2010-0031, 48th AIAA Aerospace Sciences Meeting Including the New Horizons Forum and Aerospace Exposition, 4 - 7 January 2010, Orlando, Florida.

Shope, F. L., "Conjugate Conduction-Convection Heat Transfer with High-Speed Boundary Layer," J. of Thermophysics and Heat Transfer (Vol. 8, No. 2, April-June 1994).