



## RadTherm Import from STAR-CD™

### Importing Fluid Temperatures and Convection Coefficients

#### About RadTherm Import

ThermoAnalytics' RadTherm currently uses only surface geometry. A thickness for each surface is specified in the RadTherm GUI, with 1 or 3 layers of materials composing the wall cross section. When importing geometry from other applications, it is important to remember that only surface meshes are needed in RadTherm. The method outlined here will create a Patran Neutral file with fluid temperatures and convection coefficients for import to RadTherm.

#### Importing from STAR using the posdat.f user subroutine: SETUP

**Step 1:** This user subroutine is for steady-state cases only. Users must have valid  $Y^+$  or  $Y^*$  values along the walls. This is a non-dimensional measure of the boundary layer thickness and the elements adjacent to the walls must fall within a certain range to produce valid results.

**Step 2:** All user subroutines must be placed in a "ufile" directory that is in the same location as the geometry and problem files. Windows NT/2000/XP users will have this directory automatically created at start up. Unix users must create the directory and files using the ufiles utility. To do this from the Pro-Star Interface, choose File->System Command and enter "ufiles" in the dialog box. (This command can also be executed in a UNIX-shell).

**Step 3:** Next, the post processing user subroutine must be activated. This is done by entering the Utility->User Subroutines panel from the main Pro-Star panel. The scroll list at the bottom of the window is used to select the subroutine to active. An asterisk "\*" indicates that a subroutine is activated. To activate posdat (the post processing subroutine), select it in the panel, and then press write file. Alternatively, you may simply type "prfield ,,user" in the Prostar command

The screenshot shows the 'User Subroutines' window in Pro-Star. The top pane contains the source code for the 'posdat.f' subroutine, including comments, include statements, and variable declarations. The bottom pane displays a table of subroutines with their names, Prostar commands, and descriptions.

Name	Prostar Cmd	Description
PORKEP	PORTUR	Porous-medium turbulence
POROS1	POROUS	Permeability
POROS2	POROUS	Permeability matrix
* POSDAT	PRFIELD	Post-process data
RADPRO	RADPROP	Gaseous radiation properties
RADWAL		Radiation properties at wall
REACFN	RRATE	Specification of rate of reaction
ROUGHM	RDEFINE	Define roughness parameters for a wall
SCALFN	SCPROP	Species-scalar function
* SORKEP	RSOURCE	Source-term for enthalpy
SORKEP	RSOURCE	Source-term for k-epsilon model
SORHOM	RSOURCE	Source-term for momentum
SORSCA	SCPROP	Source-term for scalar species
SPECHK	SPECIFICHEAT	Mean specific heat-capacity
TWLUSR	TLMODEL	Specify turbulence properties of two-layer model
UASI	EVSLIDE	Specify the time varying offsets for ASI Matching
UBINIT	EICOND	Specify initial conditions of included cells
UFDE2P		Define drag factor and Ct for Eulerian two-phase
UOMEGA	MFRAME	Angular velocity
UPARM	MVGRD	User-coded EGRID parameters

The screenshot shows the 'Prostar: tut' window. The 'Utility' menu is open, and the 'User Subroutines...' option is selected. The 'User Subroutines' panel is visible, showing a list of subroutines. The 'prfield' subroutine is highlighted, and the 'Write File' button is visible.

The screenshot shows the Prostar command line. The command 'prfield ,,user' has been entered. The command prompt is '1805.' and the cursor is at the end of the line.



window to activate the posdat subroutine.

**Step 4:** The final step is to copy the TAI Export subroutine into the ufile directory. This will overwrite the dummy file that Star creates.

### OPERATION

**Step 5:** Compile the Star executable as normal, specifying that you have User Subroutines to link.

**Step 6:** Solve your model. When the model has converged, a PATRAN neutral file will be output. The name of the file output is "taioutput.ntl". The file has the following information in it:

- Packet 2: element data
- Packet 1: vertex data
- Packet 17: convection coefficient
- Packet 18: fluid film temperatures

**Step 7:** Open the file "taioutput.ntl" in RadTherm. If you want to view the convection coefficient and fluid temperature data, you have two options. View it by element in RadTherm, or use TDFUtility to load the data into EnSight™ for visualization.

#### To view the CFD data in RadTherm:

- Step 1. Run the model.
- Step 2. Go to the Post Processor tab.
- Step 3. Select an element in the model.
- Step 4. Select the Boundary Conditions sub-tab.
- Step 5. Data for the selected element is displayed.

#### To view the CFD Boundary Conditions in EnSight™:

- Step 1. Open the Patran file in RadTherm.
  - Step 2. Save the file as a TDF file.
  - Step 3. Open the TDF Utility program and go to the EnSight tab.
  - Step 4. Select the TDF file
  - Step 5. Select "Both" Element Faces under the Boundary Conditions Export box.
  - Step 6. Check the "Imported Convection" box.
  - Step 7. Press the **Run** button, and select the directory where the data files for EnSight will be stored.
  - Step 8. Open up EnSight, load the Case file (\*.cas), and select the desired variables from the list below (F=Front, B=Back). F\_CFDH\_BC, F\_CFDT\_BC, B\_CFDH\_BC, B\_CFDT\_BC.
- For detailed instructions on loading and viewing boundary conditions in EnSight, see **Technical Bulletin #820**, available from the ThermoAnalytics website: [www.ThermoAnalytics.com](http://www.ThermoAnalytics.com).

