ESC TMG enhancements

TMG Open Architecture Primer

- New user documentation is available on the I-DEAS Online Help Bookshelf, the *TMG Open Architecture Primer*. This document is designed as an introduction to TMG's customization features available through user written Fortran subroutines, including integration of user supplied heat transfer correlations. Included in the Primer are best practices, coding tips and sample code. The document is intended to supplement the TMG User's Guide documentation found under Solving the Model, User Subroutines. The TMG Open Architecture Primer's table of contents appears below.

Peltier Cooler

- The new *Peltier Cooler* boundary condition allows you to simulate the specific heat loads generated by a thermoelectric heat pump.

- Values for defining a Peltier cooler boundary condition can be obtained from the device manufacturer.

Modeling with Primitives

- TMG Thermal Analysis now has a new modeling entity useful for many applications, the TMG Primitive. Creating basic 3-D geometrical arrangements of 2-D elements is much easier when working with Primitives, and makes it possible to use TMG Thermal Analysis in conjunction with other radiation analysis applications.

- Using Primitives in TMG

  - A Primitive is a TMG entity that defines elements in terms of their geometric arrangement, location, orientation, property, size, placement, element ID, element color, element order (linear or parabolic) and units of measurement. Elements that are part of a Primitive are automatically grouped on creation.

  - The *Primitives* form creates or modifies these entities by means of menu selections and text boxes. When you select a *Primitive Type* (basic shape) and a *Creation Method* (create by points or parameters), a graphic appears on the form indicating terms that define the Primitive. If you pick the graphic, a web browser window will open, with additional information for the specific Primitive Type and Creation Method depicted.

Solver Enhancements

- Faster Solution, Less Disk Space

    - Three TMG solver files are now generated in binary format rather than in ASCII as before. With the new format, MODLF, MODLCF and VUFF files are approximately 75 percent smaller, with corresponding gains in the time it takes to produce them during a solve. The size of these files sometimes caused disk space problems; the binary format resolves this issue as well.

- Solver Control Enhancements

    - Restart Control

    - You now have a choice of the directory where the TMG solver will look for the files it needs for a restart analysis. This makes for less file manipulation and fewer mistakes. Three options are provided. You can choose:

New User Interface

- TMG has a reorganized set of forms for Solver Options. The new arrangement is designed to simplify and clarify the process of setting up these options. Here are the highlights of these changes.

    - Solver Control Form

        - The Solver Control form is now the primary form for setting all options. It provides buttons to access to three commonly used sub-forms that were previously less visible, Restart Control, Initial Conditions and Advanced Solver Options. The Initial Conditions icon has been eliminated from the icon panel.

- A single button is now used to access either of the two Analysis Parameters forms, controlled by radio buttons that define the analysis as either Steady State or Transient.

    - Analysis Parameters Forms

        - The two Analysis Parameters forms (Steady State and Transient) now include buttons that access the sub-form Flow Options and the sub-form Additional Options (previously called Solution Methods). Previously these buttons were located on the Solver Control form. It is far more likely that these less common options will be selected after completing the selections on the analysis parameters form.

Radiation Enhancements

- Matrix Simplification for Large Radiation Models

  - A new option on the Advanced Radiation Parameters form, *Use Radiation Patches*, can significantly reduce solution time on large models with complex radiative exchange. The option works by temporarily merging adjacent elements for the purposes of calculating radiative exchange. After the calculation of view factors for individual elements, Oppenheim elements that are coplanar within a 15 degree tolerance are treated as a single element in the radiative exchange matrix.

  - Before merging elements, TMG performs several checks to ensure that merging is appropriate. In addition to being adjacent and coplanar within a tolerance, the elements must have identical radiative material properties, and must not be separated by any surface that intersects their surface (such as the two arms of a "T" intersection.)

- Infrared Specularity on Reverse Sides Entities

  - When creating a Reverse Sides entity, you can define the IR specularity to be the same as the front side properties of each element, or you can specify a new value for the entity. Previously, the only IR property available for this purpose was emissivity.

- Infrared Spectrum Radiative Heat Source

  - You can now define heat source elements as radiating either solar or infrared spectrum energy. Previously, only solar spectrum was available. Heat flux view factors from the source elements to the illuminated elements are calculated. You can control the accuracy of the shadowing calculations by selecting an Error Criterion value. You must define emissivity for infrared Radiative Heating.

Articulation Enhancements

- Orbit Dependent Articulation
    - A rotational articulation can now be linked to defined orbits. This is accomplished simply by selecting a vector on the part and selecting the desired orbit dependent direction:
        - Nadir
        - Sun
        - Star
        - Velocity Vector
        - Orbit Normal
        - North
        - South
        - If, because of a conflict with a vector defined on the Spacecraft Attitude form, it is not geometrically possible to perform the orbit dependent articulation specified, TMG will articulate the member so as to minimize the discrepancy.
- Visualizing Orbital Articulation
    - The motion of articulated members of spacecraft can now be viewed using the Orbit Visualizer. The animated FE model in orbit accurately displays all the rotations and translations of its articulated parts, including compound articulations and articulations that track an orbit parameter.
- Removal of Articulation Limitations
    - The Articulation algorithm has been enhanced to be compatible with the following TMG features, formerly not compatible:
- Hemicube view factor calculation method
- Ray tracing
- All radiative heating
- Axisymmetric models
- All radiation calculation methods and residual view factor options

Freeze Flow Field Solver Option

- A new Flow Solver option can dramatically reduce solution times for problems where flow conditions are not strongly dependant on temperature. The option is called *Freeze Flow Field Conditions* and, if selected, suspends certain flow calculations when they remain relatively constant from one iteration to the next. This allows the thermal solver to continue to iterate under the assumption that the flow will be unaffected. If the temperature change between iterations is too great, the flow solver returns to its normal mode to verify that flow conditions are still unchanged. In this way, the flow solver is prevented from performing repetitive unnecessary calculations.

- In the freeze flow mode, the flow solver stops solving the equations for velocity and mass, but continues solving the energy equations for flow.

- The option is available on the Flow Solver Additional Options form. The criteria for switching from the normal flow solver mode to the Freeze Flow Conditions mode are specified in terms of maximum values for velocity change and pressure change for transient problems, and in terms of flow convergence for steady state problems. The criteria for switching back to the normal flow solver mode are specified in terms of temperature change.

- By using this option, solution time can be greatly reduced in many forced flow analyses. It should not be used when buoyancy and natural convection are a significant factor, or when fluid material properties are temperature dependent.

Humidity, Evaporation and Condensation

- Humidity and Other Fluid Mixtures

  - You can now model the mixing of water vapor in air, and/or of another component in the fluid domain such as a second fluid, dust or smoke. These features, introduced in a limited way through generic entities in I-DEAS 9, are now fully integrated into the user interface.

  - Water vapor and/or other fluid or fluid-like components (here called "the second fluid") are introduced into the fluid domain using (inlet) Fans or Vents. The second fluid is introduced at these points, and is carried through the fluid domain. Using these entities, or the related entity Ambient Conditions, you can model the mixing, in any proportion, of the second fluid in the main fluid. It can be introduced at the same level through all inlets to the fluid domain by specifying the value on the Ambient Conditions form, or uniquely at each inlet by specifying the value on each Fan or Vent form. Humidity introduced in this way can be specified from the form in relative or specific terms.

  - You can also specify Humidity or other fluid mixtures as an Initial Condition for all fluid elements.

- Modeling Evaporation and Condensation

  - In a transient analysis, you can model evaporation and condensation of water on Flow Surfaces when the ambient fluid is air. You can define humidity as an Initial Condition of all air elements, or introduce humidity into drier air through Vents or Fans. Heat loads and cooling by evaporation and condensation are accurately modeled on Flow Surfaces. Results in terms of water density on elements for each result output time are available for post processing.

  - You can specify the initial water content on a Flow Surface by creating a Generic Entity for this purpose. The flow solver treats this as an initial condition at the start time of the transient analysis. Water content on the Flow Surface is modeled as a

Periodicity Boundary Condition Enhancements

- The Periodicity Boundary Condition, introduced in I-DEAS 9, has been upgraded and enhanced for I-DEAS 10. An icon and a form-based entity specification interface are the primary enhancements.

- This new feature has has the potential of greatly reducing model size and solution time in certain applications. If the application can be modeled as a series of identical joined sections, you need construct only one section. Periodic Flow boundary conditions will force the flow and scalar fields to be identical at the periodic boundaries, resulting in a spatially cyclic solution. The sections can be identical translationally or rotationally. The periodicity is both fluid and thermal.

- Periodicity can only be used if the entire model can be broken down into identical periodic sections. Each periodic section must have a pair of matching and opposed periodic faces with identical fluid/thermal conditions on the face. The two periodic faces must have identical meshes. A typical translational example would be a series of identical baffles. A typical rotational example would be flow in a cylinder.

- Periodic Flow boundary conditions are used to simulate a flow leaving through a Boundary A and entering through a Boundary B (or vice versa) under identical conditions (velocity, temperature, scalar values, etc...). The periodic boundary conditions act as if the solution domain were rolled up so that boundaries A and B were adajacent. As shown in the simple examples below, only the segment between Boundary A and Boundary B need be modeled. The other segments would produce identical results. Hence, these models are appropriate applications for a Periodic Flow boundary condition.
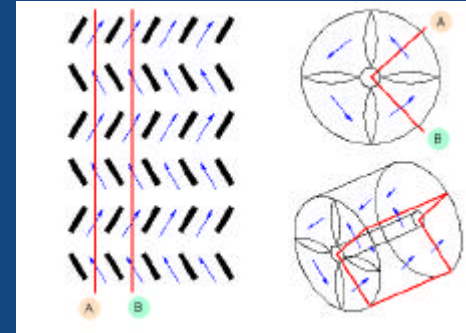
Velocity-proportional Head Loss in Screens

- A new option introduced as a hidden feature in I-DEAS 9m2 has been upgraded and integrated with the ESC user interface for I-DEAS 10. The feature allows you to model the pressure drop through a Screen as proportional to the approach velocity of the fluid. Previously, head loss was always calculated as proportional to the dynamic pressure. This may not be suitable in some practical cases, such as flow through relatively thick screens.

- The option can now be selected on the Screen form where it is specified as a coefficient f representing the proportionality ratio and having the units of a mass flux (P = -f*V).

Rotating Frame of Reference

- A Rotating Frame of Reference is an ESC entity that allows you to find a steady state flow solution for objects spinning at a constant rate about an axis, within a fluid. At the interface between a rotating frame of reference and a static frame of reference, physical fluid flow is modeled correctly. You can include one Rotating Frame of Reference in your model, or many.

- Using a rotating frame of reference greatly reduces the computational load that would be incurred if the same problem were to be solved as a transient analysis explicitly modeling the rotation of the object in the fluid. Instead, the flow solver assumes that the 3-D fluid elements are rotating about the same axis, while the object is stationary. From a CFD standpoint, this assumption is equivalent to the physical case, and the spinning fluid is the frame of reference for the flow solution. The flow solver calculates how the "stationary" object impinges on the velocity of the spinning fluid in a steady state condition. Element velocity results are provided in terms of how the steady state velocity of each element differs from the rotational velocity of the frame of reference.

- By defining Flow Surfaces you can accurately model convection on the surfaces of the rotating object as well as on surfaces that border the frame of reference.

- ESC allows for several rotating frames within a given model. You can also combine rotating frames with an inertial frame. The frame change conditions on velocity are implicitly handled by ESC.

- Some Rotating Frame of Reference Applications

- RFR comprises entire model, all surfaces spinning.

- RFR comprises the entire model, cylinder wall is stationary, spinning surfaces are attached to a shaft that forms the axis of the cylinder.

- RFR is located within a larger fluid volume that forms an intertial frame of reference.
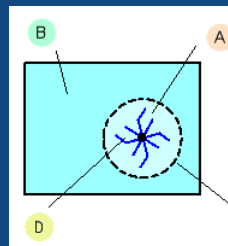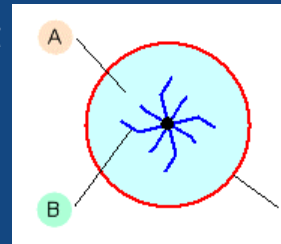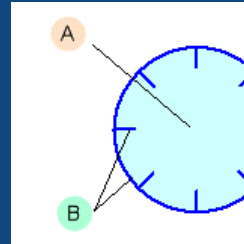
Multiple Fluids

- ESC now supports models with multiple fluids in separate domains. Any number of different fluids can be used in the same model, provided each fluid is contained in a separate volume and there is no mixing or exchange between the different fluids. Each volume is considered a separate fluid domain. Heat may be exchanged between different fluids through convection on a Flow Surface that separates the two fluids, or by thermal couplings or conduction between two separate Flow Surfaces.

- Fluid material properties are defined on the 3-D elements as always. A fluid that enters a volume from the ambient environment (via a Fan or Vent) is assumed to have the same properties as the fluid in the volume it is entering. By default, all fluids entering the different volumes are at the same Pressure, Temperature, Humidity etc., as defined in Ambient Conditions. If you want to specify these conditions differently, you can do so by defining them on the Vent or Fan where the fluid enters.

- All the solver controls and options apply uniformly to all the volumes containing different fluids. There is no way, for example, to use a different viscosity model for different fluids.