

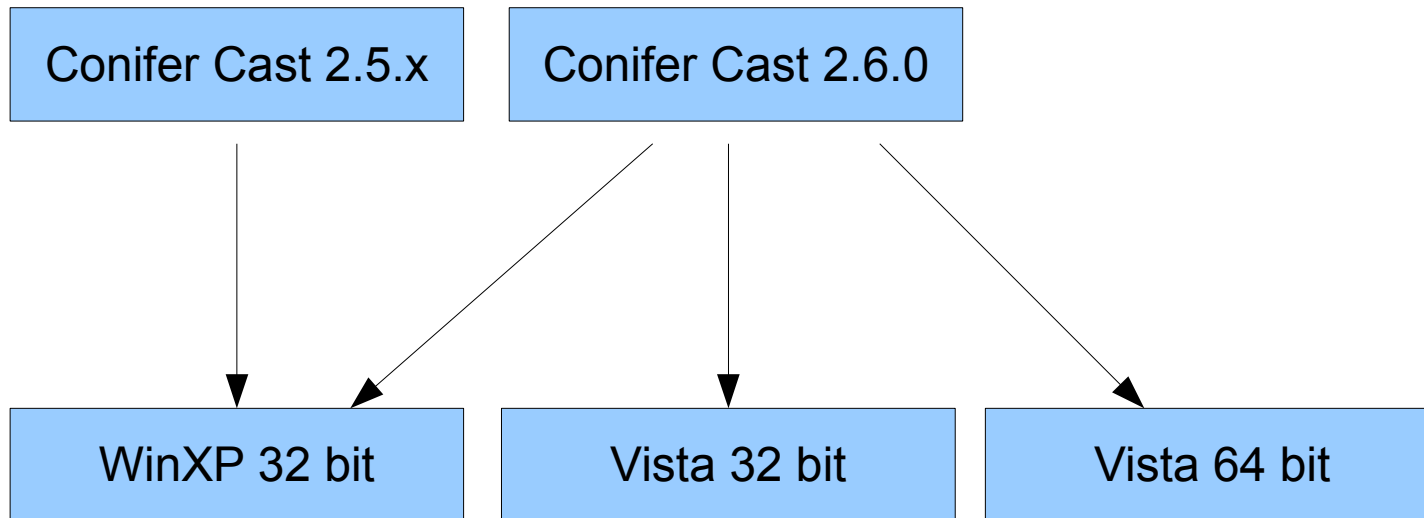
Conifer Cast 2.6.0

New features

- Windows Vista Compatibility
- New Installer
- Solver Pause/Resume
- Locally Implicit Advection
- Split Lagrangian VOF Advection
- Domain Removing Components
- Unstructured Memory Allocation (UMA)
- XML Export/Import of Material Database
- Limited Compressability
- Heat Transfer Coefficients to Mesh Border

Windows Vista Compatibility

- Conifer Cast 2.6.0 is a 32bit Windows Application
- Conifer Cast 2.5.x supported
 - Windows XP SP2 (32bit)
- Conifer Cast 2.6.0 adds support for
 - Windows Vista (32bit)
 - Windows Vista (64bit)
- Both MAC address based and dongle bound licenses are possible



New Installer

Before

FLOW-3D Installer

Conifer Cast Installer

Dongle Driver Installer

Restricted Version
Solver Executables

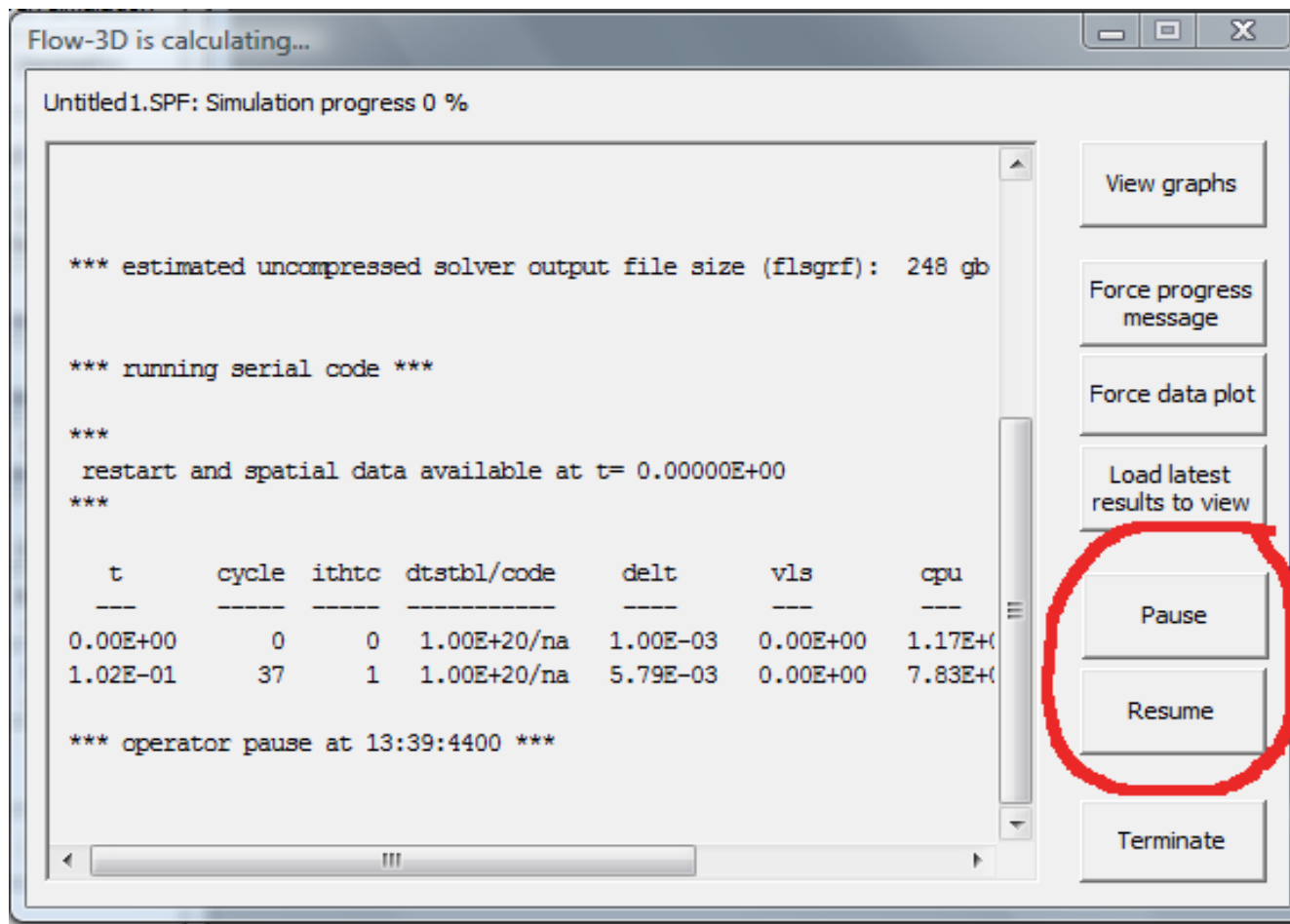
FLOW-3D License
Configuration

Now

Conifer Cast Installer

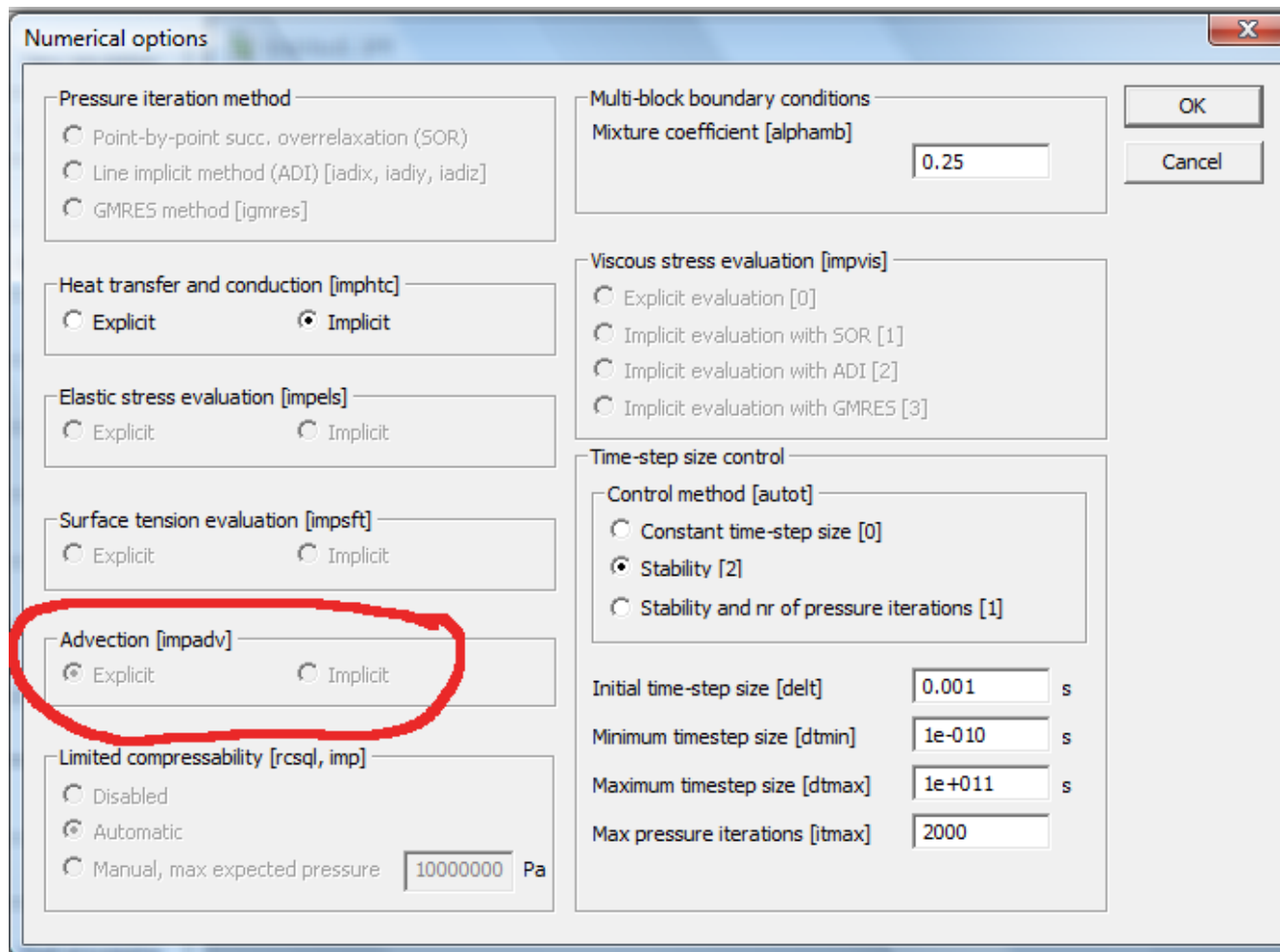
- In Conifer Cast 2.5.x installing was difficult
- In Conifer Cast 2.6.0 a new installer has been introduced which simplifies installation considerably
- Installer installs
 - Conifer Cast user interface
 - FLOW-3D Solver
 - FLOW-3D User Interface
 - FLOW-3D License Service and License File
 - Manuals as PDF files
 - USB Dongle driver for dongle bound licenses
- Installer configures
 - Language version
 - Limited version solver executable according to license

Solver Pause/Resume



- During simulation it is possible to put the solver into pause mode
- Pausing solver will temporarily release the license allowing another simulation to be run
- Pausing solver will NOT release the memory occupied by the solver

Locally implicit advection 1/2



- New feature in FLOW-9.2 solver
- Enabled from Numerical options

Locally implicit advection 2/2

Designed to relax the Courant stability limit on the time step size for faster calculations:

$$\Delta t < CON \Delta x / U, \quad CON = 0.45$$

- Applies selectively when and where the Courant condition is violated. Otherwise, an explicit method is used for better accuracy
- Fast and efficient, no iterations required. Adds ~5% CPU time per time step and memory
- Accuracy is maintained by controlling the time step size and by selective application
- Applies to all advection equations: momentum, fluid fraction, energy, *etc.*
- Not used near free surface and two-fluid interface to maintain accuracy of the VOF method – fluid fraction is always advected *explicitly* near free surface

Split Lagrangian VOF advection

Advanced options

Viscosity [ifvis]

- Local viscosity evaluation (no turbulence model) [0]
- Two-equation turbulence model [3]
- Renormalized Group Theory model [4]

Density [ifrho]

- Constant uniform density [0]
- Function of local temperature [1]

Drag function [idrg]

- No drag [0]
- Volume fraction dependent drag [1]
- Fluid fraction dependent drag [2]
- Unsaturated flow in porous media [5]
- Reynolds number dependent drag [4]

Advection option [ifvof]

- VOF + Interior Donor Cell [4]
- Unsplit Lagrangian method [5]
- Split Lagrangian method [6]

Other models

- Disable wall shear stress calculation [iwsh]
- Enable cavitation mode [icav]
- Enable viscous heating [ivish]
- Enable surface tension [ifsft]
- Enable wall adhesion [cangle]
- Enable surface defect tracking [idfct]
- Enable micro-porosity evaluation [mupor]
- Enable air entrainment evaluation [idfair]
- Shear dependent viscosity model [muc1]
- Temperature dependent viscosity model [mutmp1]

Gas model [gamma]

- Fixed pressure gas regions
- Adiabatic gas regions

Adiabatic constant: 1.41

Gravity [gx, gy, gz]

X: 0 Y: 0 Z: -9.81 m/s²

Restart

Restart time [trest]: 0

Temperature options [ihtrst]

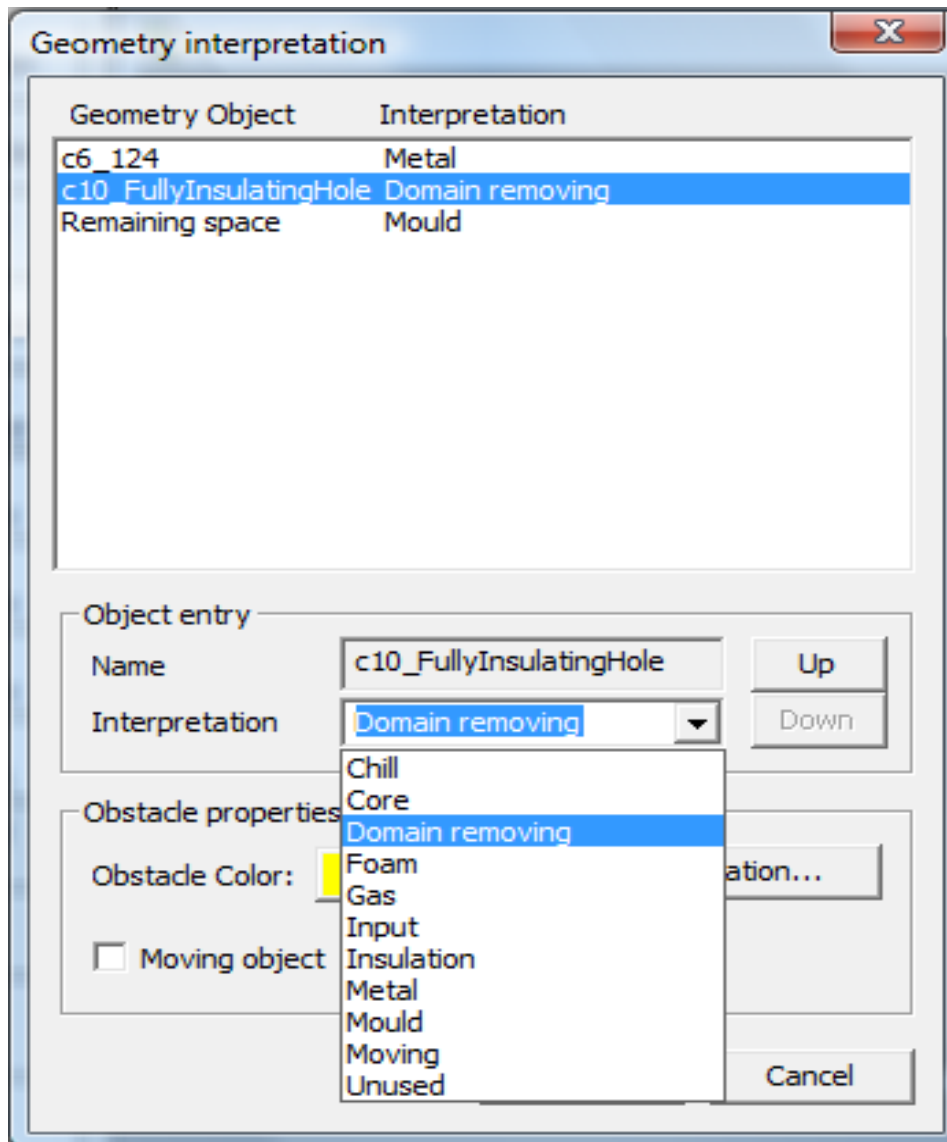
- Use all data from previous run [0]
- All data except temperatures [1]
- Solid temperatures only [2]

OK

Cancel

- New model introduced in FLOW-3D 9.1 solver
- The Split Lagrangian method typically produces lower cumulative volume error than the other methods in FLOW-3D, although the volume error may increase when this method is used together with the GMO model.

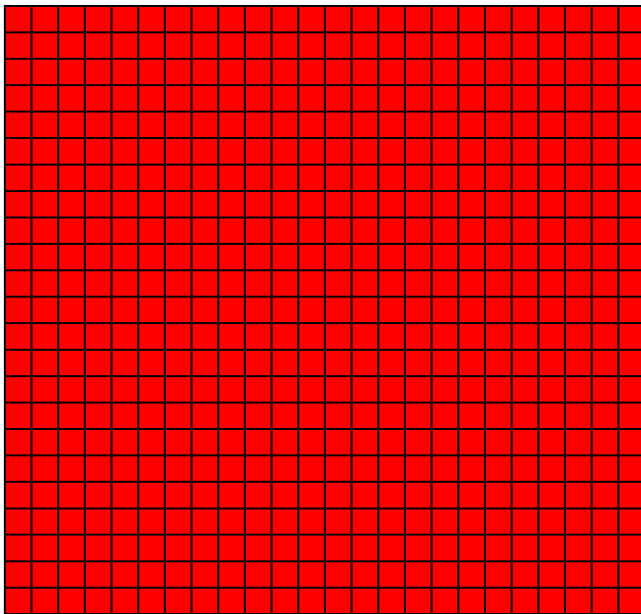
Domain-removing components



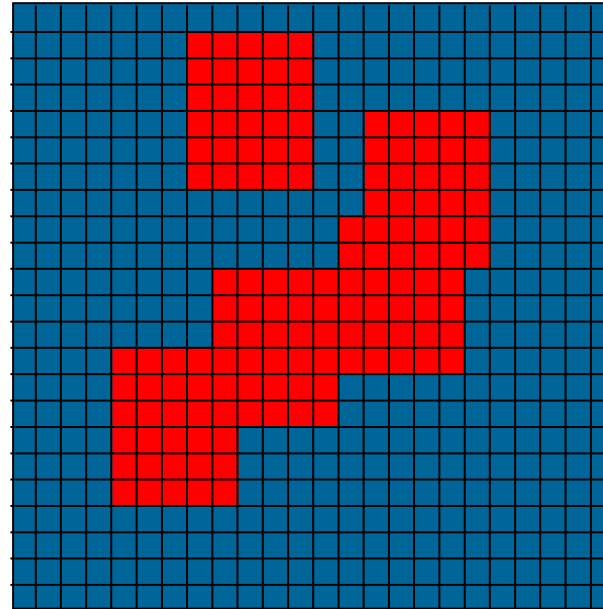
- New geometry interpretation type of 'Domain removing' has been added to Geometry interpretation dialog
- Setting an STL as domain removing will mark the area occupied by the STL as inactive
- No calculation whatsoever happens in the inactive area. No calculation elements are generated for the area.
- Can be used for removing areas from calculation domain that do not have any impact for the solution in order to gain better calculation performance.

Unstructured Memory Allocation (UMA)

Structured meshing:
all cells are in memory

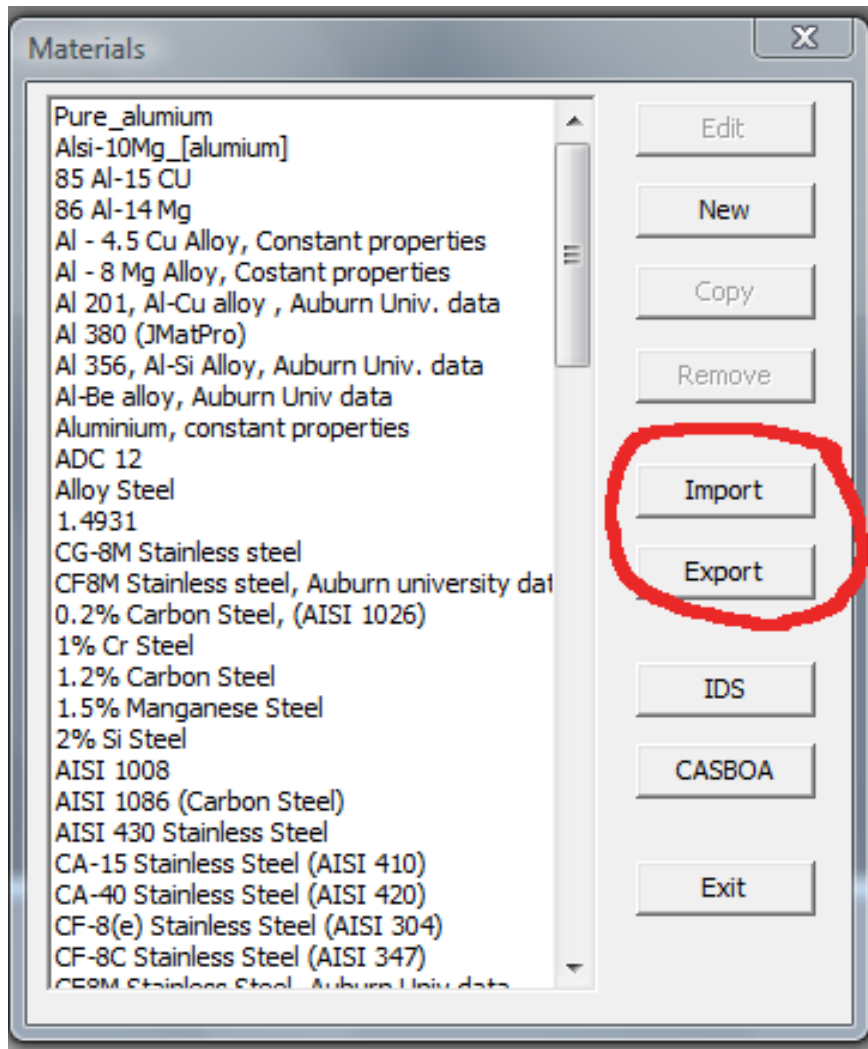


Unstructured meshing:
only *active* cells are in memory



- FLOW-3D 9.2 solver feature
- Ability to allocate solution arrays on parts of the mesh for faster more efficient calculations
- Benefits
 - Shorter arrays
 - Less memory use
 - Faster execution
- Automatically enabled, no user action needed
- Big impact on performance of flow-only simulations (i.e. no heat transfer)

Export/Import to XML of material database



- The contents of material database can be exported to an XML file
- Material data in XML format can be imported in to material database
- Usefull for moving material data to/from 3rd party software
- Usefull for taking backups of material data
- Usefull for merging two material databases

Limited Compressability

Numerical options

Pressure iteration method

- Point-by-point succ. overrelaxation (SOR)
- Line implicit method (ADI) [iadx, iadiy, iadz]
- GMRES method [igmres]

Multi-block boundary conditions

Mixture coefficient [alphamb]

Heat transfer and conduction [imphtc]

- Explicit
- Implicit

Elastic stress evaluation [impels]

- Explicit
- Implicit

Surface tension evaluation [impsft]

- Explicit
- Implicit

Advection [impadv]

- Explicit
- Implicit

Limited compressability [rcsq, imp]

- Disabled
- Automatic
- Manual, max expected pressure Pa

Viscous stress evaluation [impvis]

- Explicit evaluation [0]
- Implicit evaluation with SOR [1]
- Implicit evaluation with ADI [2]
- Implicit evaluation with GMRES [3]

Time-step size control

Control method [autot]

- Constant time-step size [0]
- Stability [2]
- Stability and nr of pressure iterations [1]

Initial time-step size [delt] s

Minimum timestep size [dtmin] s

Maximum timestep size [dtmax] s

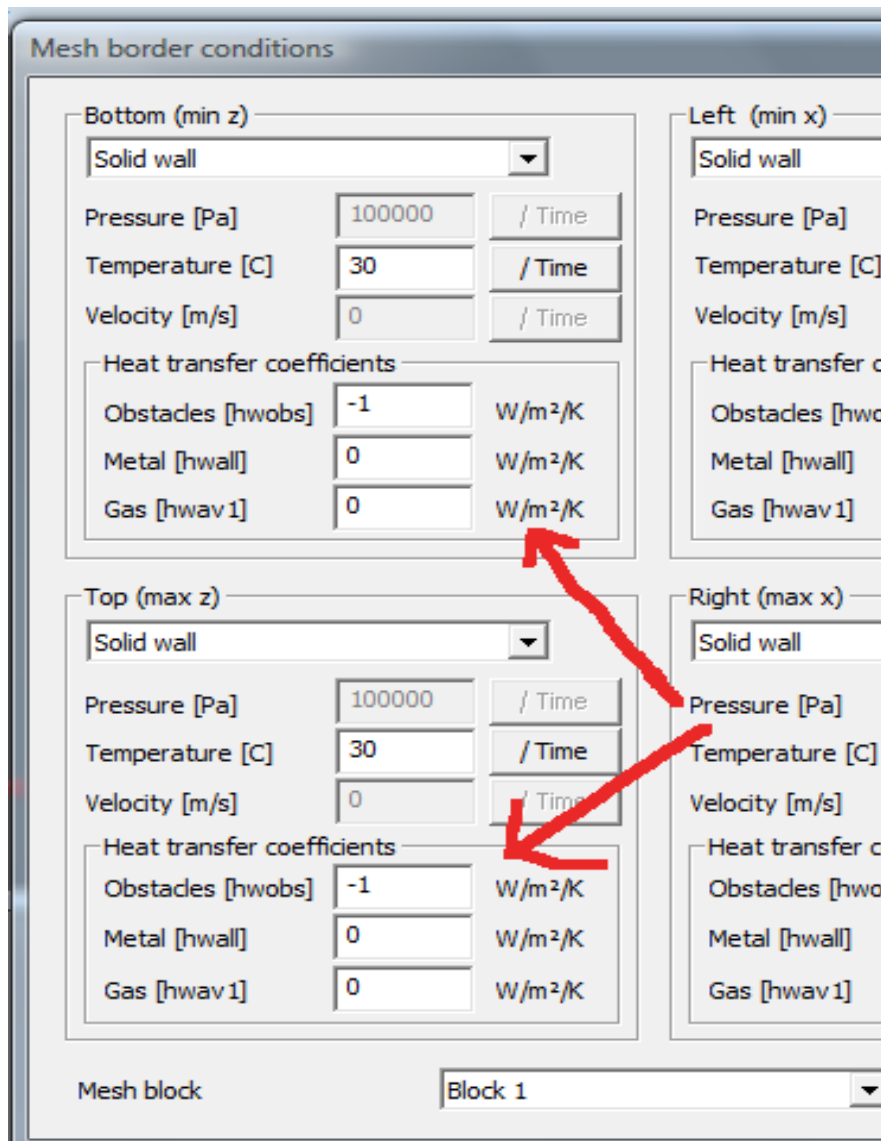
Max pressure iterations [itmax]

OK

Cancel

- Introduces very small limited numerical compressability to the metal
- Helps convergence in difficult flow simulations while not having any significant impact on the results
- Automatic limited compressability requires HPDC or Advanced license

HTC from obstacle to border in border conditions



- Border conditions dialog now allows setting heat transfer coefficient against mesh border for:
 - Metal
 - Obstacles