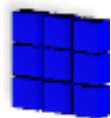


CFD-ACE+ V2020.0

Release Note

(주) 경원이앤씨



CFD-GEOM V2020.0

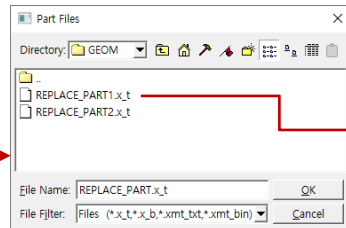
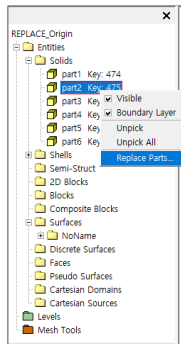
◆ CAD 파일로 부터 Solid 부분 변경

➤ 지원 가능한 CAD 파일

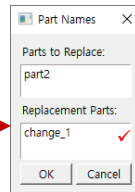
SAT, Parasolid, STEP, UG/NX, SolidWorks, ProE, Catia V5.

실행 방법

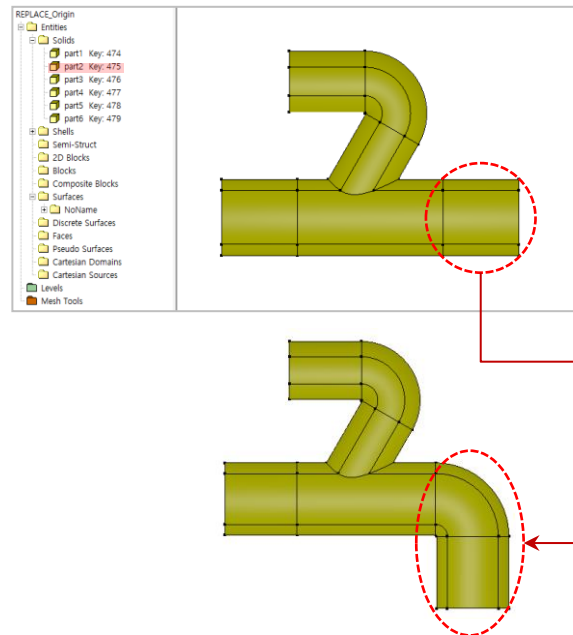
1. Model Manager(Ctrl+M)의 Solids 항목에서 원하는 파트 선택
※ 각 Solids의 파트 이름은 'Noname'이 아닌 각각 다른 이름으로 설정
2. 마우스 오른쪽 버튼 클릭 후 Replace Parts 선택

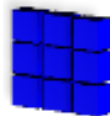


※ GGD 파일도 가능



클릭 후 OK





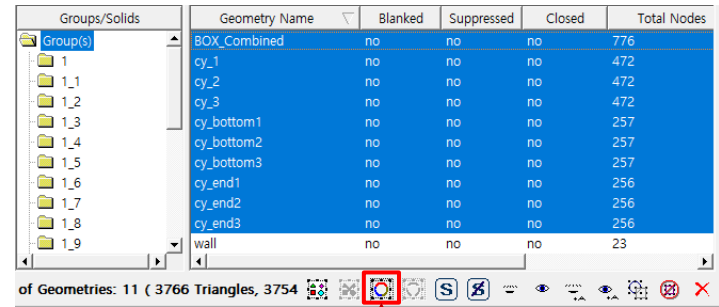
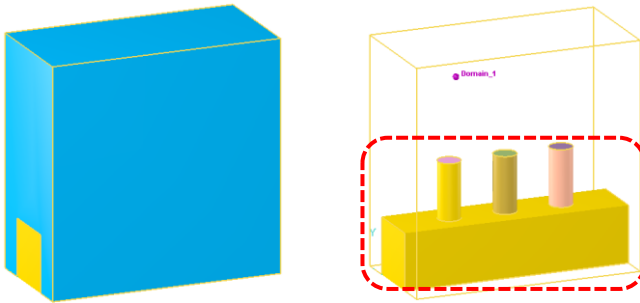
CFD-VisCART V2020.0

◆ Cartesian & Tetrahedral HYBRID 격자 생성

실행 방법

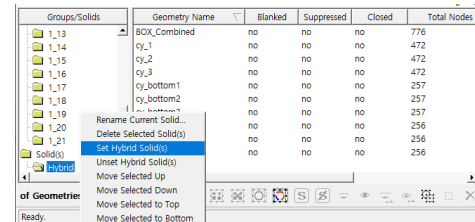
1. Create/Import Solids

※ 전체 형상에서 Tetrahedral 격자가 생성 될 부분 선택 후 'Add the selected item(s) to a solid' 버튼 클릭



2. 선택 된 Solid(s) 그룹에서 마우스 오른쪽 버튼 클릭

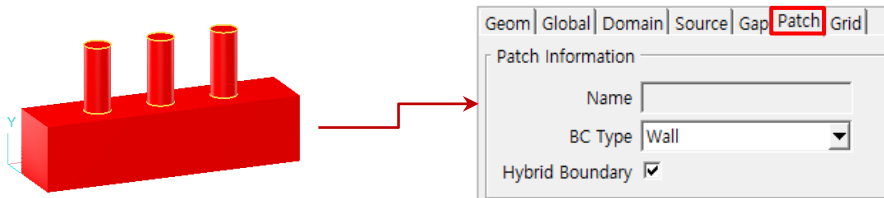
Set Hybrid Solid(s)



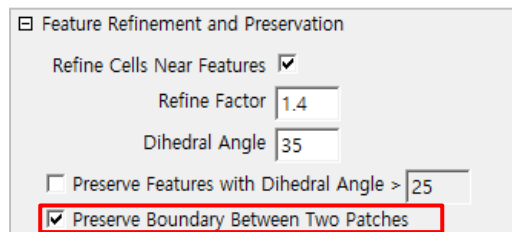
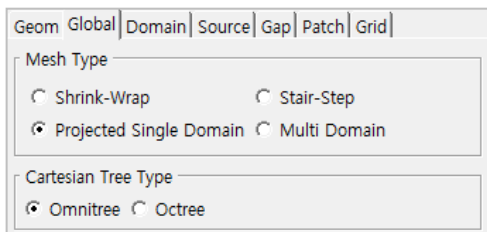
◆ Cartesian & Tetrahedral HYBRID 격자 생성

실행 방법

3. Solid(s) 그룹 요소들을 선택하고 Patch 탭에서 Hybrid Boundary 체크



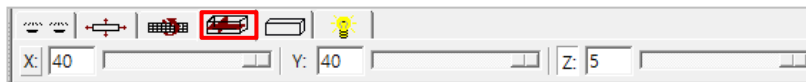
4. Global 탭에서 Mesh Cell Size 및 'Preserve Boundary Between Two Patches' 설정
Hybrid 격자 생성의 경우 'Projected Single Domain' Mesh Type에서만 실행 가능



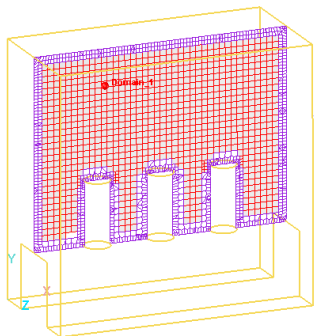
◆ Cartesian & Tetrahedral HYBRID 격자 생성

실행 방법

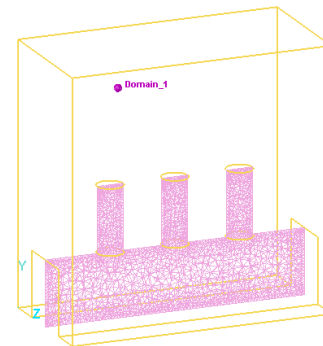
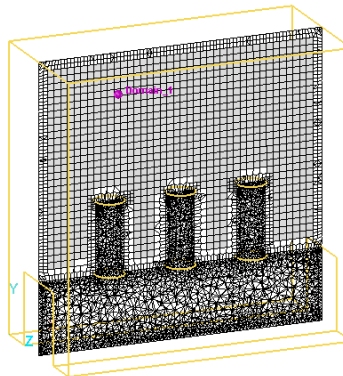
5. Generate Mesh 실행 후 Cutting Plan을 통해 확인

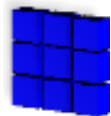


6. Domain 탭에서 각 항목을 선택하여 Domain 영역별 격자 확인 가능



Domain Name	Type	Suppressed	Property	No. Surfaces	No. Cells
Domain_1	Marker	no	Fluid	11	94261
Hybrid	Marker	no	Solid	0	230817

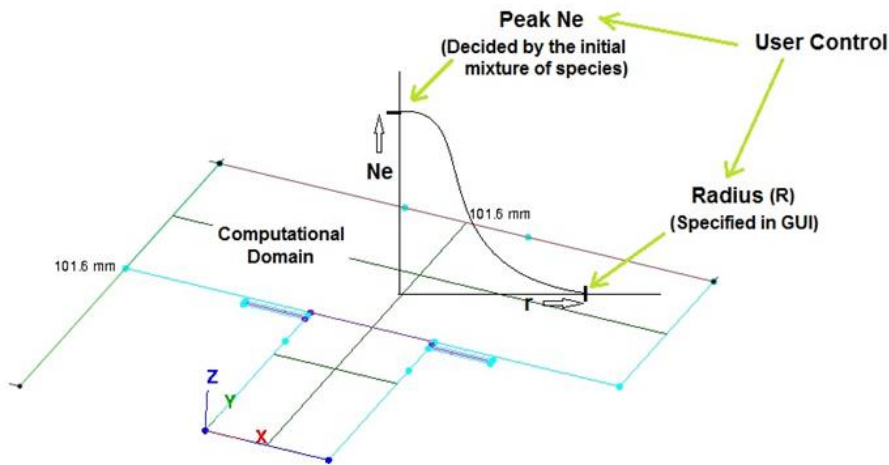




CFD-ACE+ V2020.0

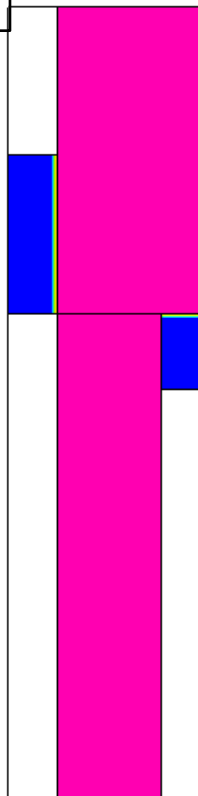
◆ Gaussian plasma density 초기값 설정

- Electron number Density의 초기 조건을 설정할 때 Quasi-neutrality 조건을 이용하여 초기 species mixture 에 의해 Peak number density 계산되고 charged particles(electrons and ions)을 Gaussian 분포로 설정 가능
 - 초기의 Peak number density가 최종 값과 같을 때 총 계산시간이 줄어 들며 iteration 수 감소
- ICP + CCP 해석에서 코일 근처 또는 CCP 해석에서 powered electrode 근처에 높은 초기 electron number density를 설정 할 경우 발산할 수 있음



PT	MO	VC	BC	VR	IC	SC	Out	Viz	Run
IC Setting Mode									
Initial Condition <input type="text" value="User Specified"/>									
IC Applied <input type="text" value="For All Volumes"/>									
Shared									
Flow									
Chem									
Plasma									
Electr									
Magn									
Electron Temperature									
Te <input type="text" value="1.00000001658621"/> J									
Electron Number Density									
<input type="text" value="Gaussian Distribution"/>									
Center X <input type="text" value="0"/> m									
Center Y <input type="text" value="0"/> m									
Center Z <input type="text" value="0"/> m									
Decay Radius <input type="text" value="0.1"/> m									

V2019.0



IC Setting Mode

Initial Condition

IC Applied

Shared

Flow

Chem

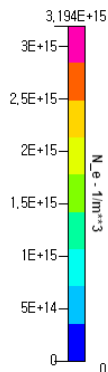
Plasma

Electr

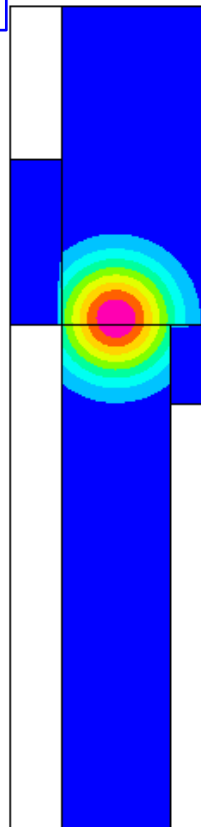
Electron Temperature

Te eV

Electron Number Density



V2020.0



IC Setting Mode

Initial Condition

IC Applied

Shared

Flow

Chem

Plasma

Electr

Electron Temperature

Te eV

Electron Number Density

Center X cm

Center Y cm

Center Z cm

Decay Radius cm

Typical low pressure CCP density is of $10^{16}/\text{m}^3$.
Electron density by quasi-neutrality:

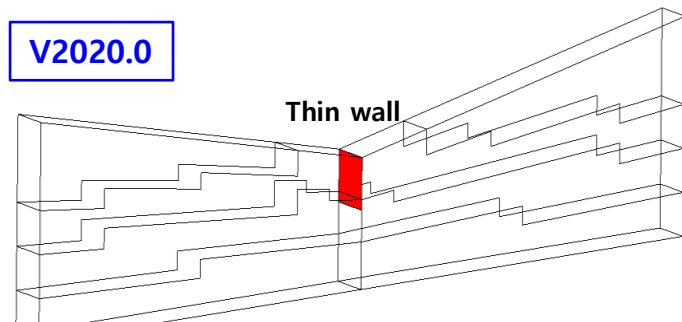
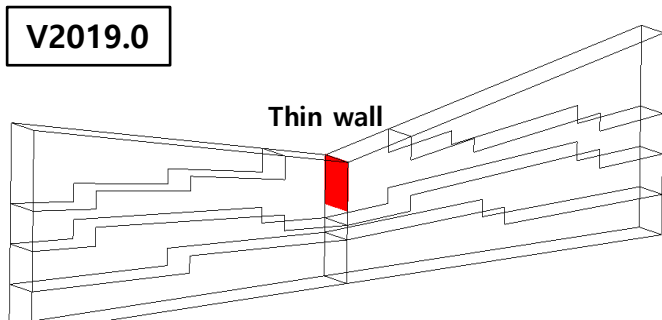
$$N_e = 6.023 \times 10^{26} \sum_{i=\text{ions}} q_i \frac{\rho Y_i}{M_i}$$

and

$$\rho = \frac{p}{(R_u \sum_{i=\text{all}} \frac{Y_i}{M_i}) T}$$

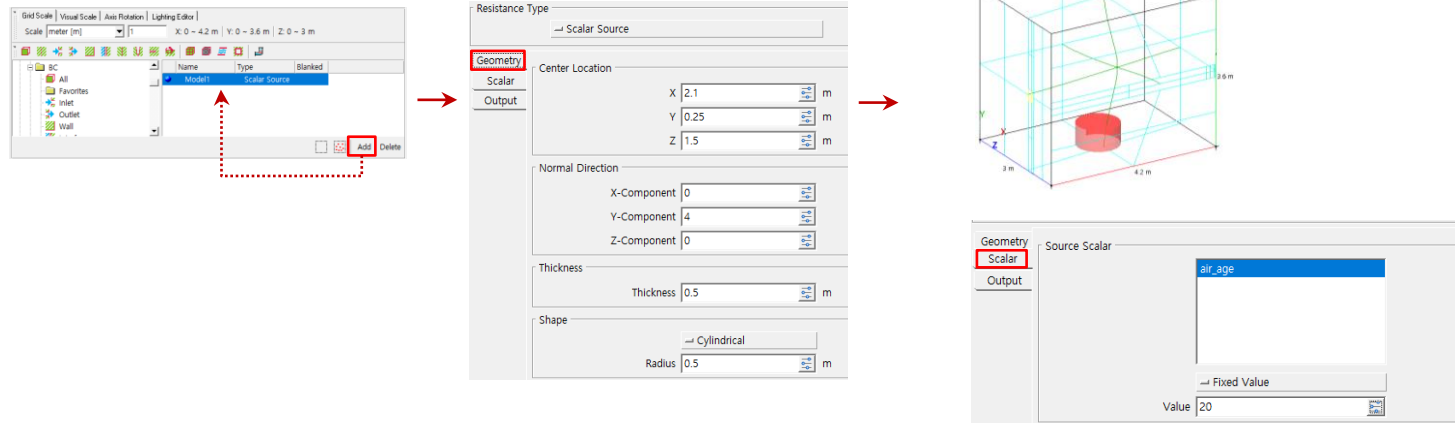
◆ Split interface boundary condition 병렬 처리 가능

- Split interface 경계조건은 Interface에서 두개의 경계조건 사용 가능 (Ex> wall and inlet/outlet)
- 이전 버전에서는 Split interface 경계조건이 있는 경우 병렬 처리 불가능 → V2020.0에서는 병렬처리 가능
 - 도메인을 분할 때 dtf_decompose 명령어와 함께 thinWallCuts=perpend 옵션을 사용함
- Thin wall의 경우도 병렬 처리 가능 → 로드 밸런싱 개선 및 병렬 기능 향상



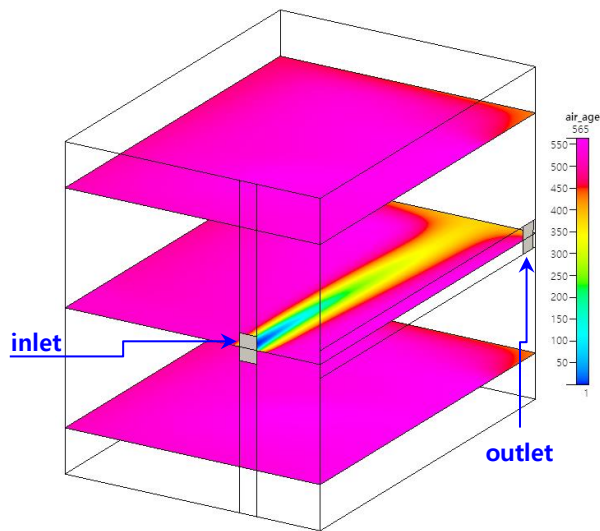
Virtual Resistance 탭에서 Scalar source 적용 가능

- Volume condition으로 지정하지 않고 사용자가 형상 지정 가능
- Porous media, momentum resistance, Scalar source에 대한 조건 설정 가능
- 실행방법
 - PT – User Scalar(Scalar) 선택 → **PT** | MO | VC | BC | VR | IC | SC | Out | Viz | Run
 - VR – Add버튼 클릭 – Geometry 탭에서 형상 설정
 - VR – Scalar 탭에서 각 조건에 맞는 source를 적용

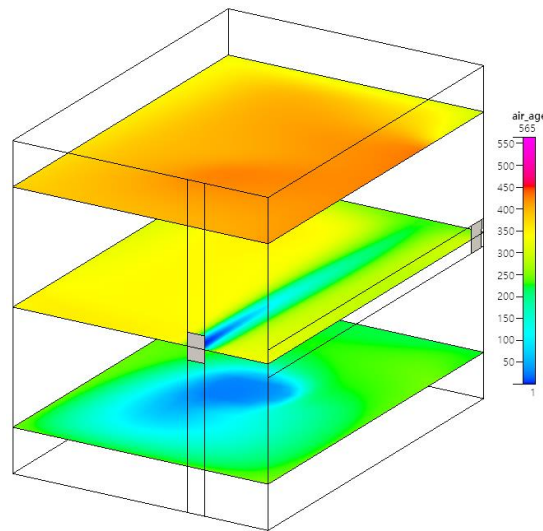


Virtual Resistance 탭에서 Scalar source 적용 가능

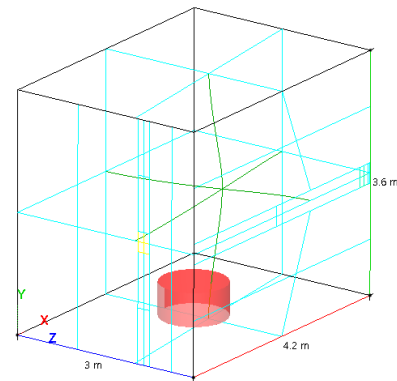
- Volume condition으로 지정하지 않고 사용자가 형상 지정 가능
 - Cylindrical, Polyhedral 선택 가능



Scalar source 적용 전

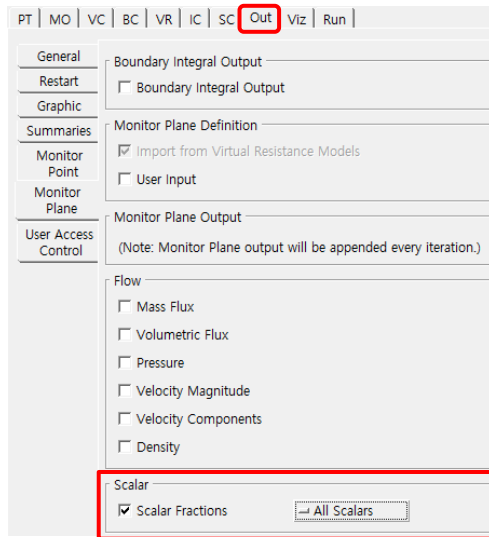


Scalar source 적용 후

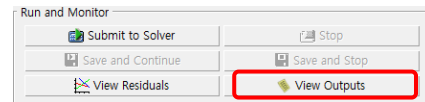
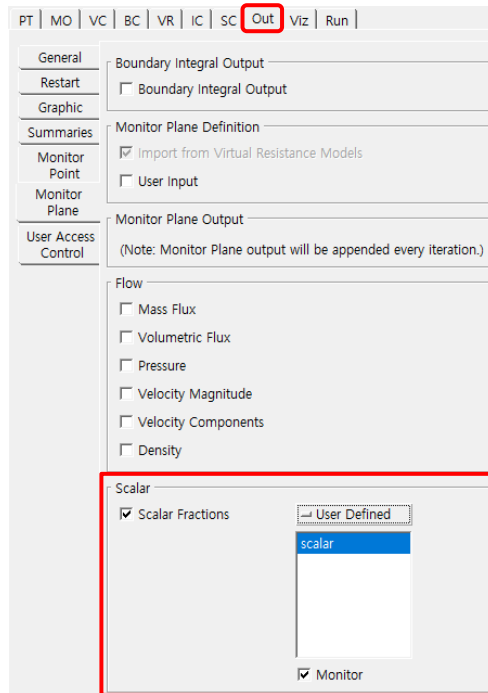


◆ 사용자가 지정한 평면에서 User Scalar 값 monitoring 가능

- 모든 User Scalar 값 또는 사용자가 지정한 User Scalar 값만 monitoring 가능



or




View Outputs 내용을 통해 확인 가능

◆ 계산 도중 User scalar의 linear solver sweep의 수와 수렴 기준 수정 가능

➤ 실행 방법 : Tools > Create MOD file

Keyword	Data	Purpose
Numeric_Scalar_Solvers_Sweeps_ScalarName	Integer	Changes number of sweeps for Scalar Variable Solver
Numeric_Scalar_Solvers_Criterion_ScalarName	Real Value	Changes the solver convergence criterion for Scalar variable

 Change the values and press 'Generate MOD File' when you are done.

Solver Control\Numeric Solver Commands

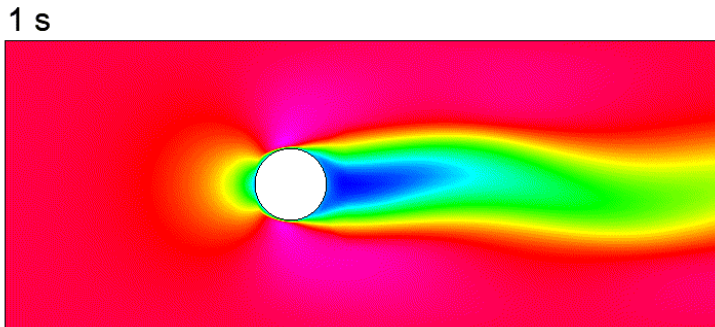
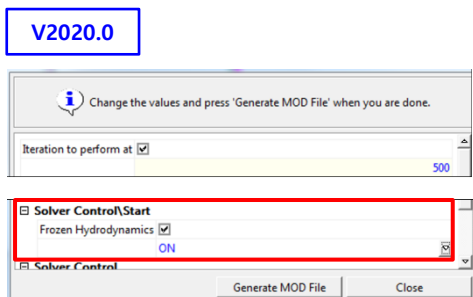
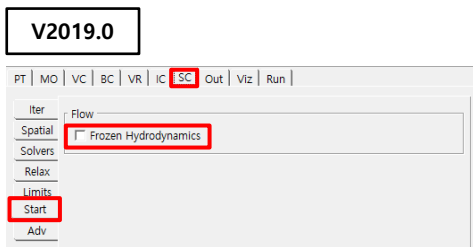
Enthalpy	<input type="checkbox"/>
Enthalpy Sweeps	<input type="checkbox"/>
Enthalpy Criterion	<input type="checkbox"/>
scalar	<input type="checkbox"/>
scalar Sweeps	<input checked="" type="checkbox"/>
	50
scalar Criterion	<input checked="" type="checkbox"/>
	0.1

Solver Control

Generate MOD File Close

◆ MOD file을 이용하여 계산 중 유동 계산을 멈추거나 시작하는 기능 추가

- 이전 버전에서는 SC > Start에서 Frozen Hydrodynamics 선택 후 진행 가능
→ V2020.0 에서는 MOD file을 이용하여 해석 중에 Frozen Hydrodynamics ON/OFF를 사용할 수 있음
- 실행 방법 : menu - Tool > create MOD file 클릭



Frozen Hydrodynamics ON : 30~80초 구간