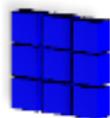


CFD-ACE+ V2020.0

Release Note

(주) 경원이앤씨



CFD-GEOM V2020.0

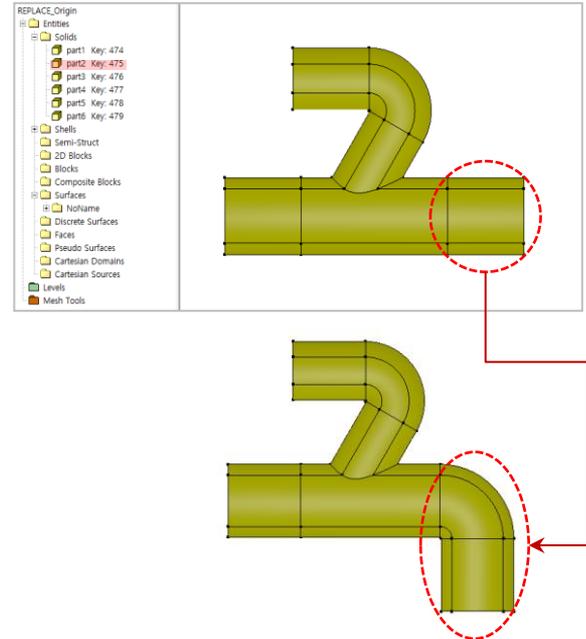
◆ CAD 파일로 부터 Solid 부분 변경

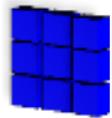
➤ 지원 가능한 CAD 파일

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





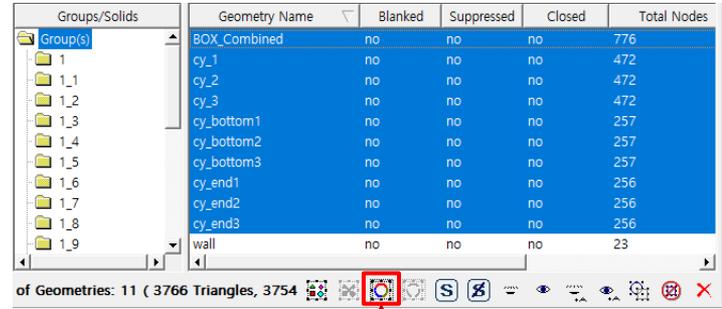
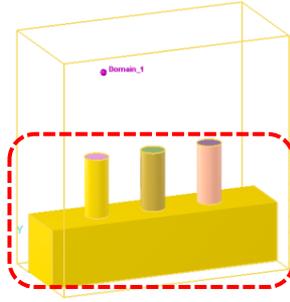
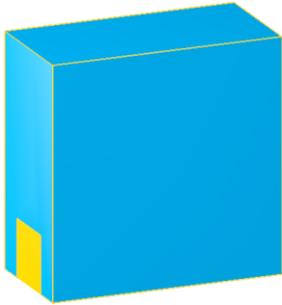
CFD-VisCART V2020.0

◆ Cartesian & Tetrahedral HYBRID 격자 생성

실행 방법

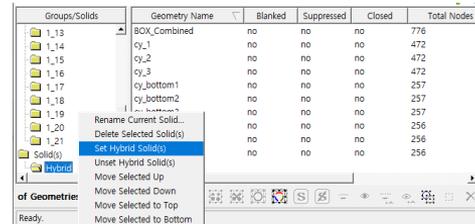
1. Create/Import Solids

※ 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

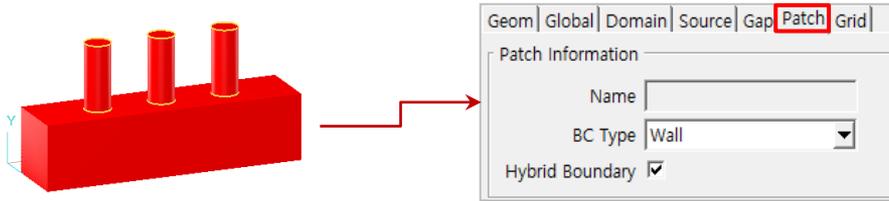
Set Hybrid Solid(s)



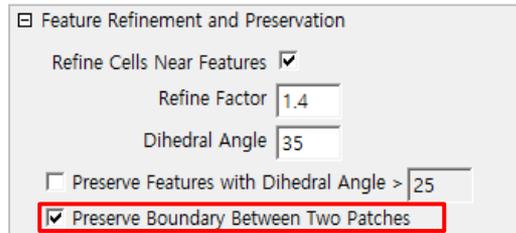
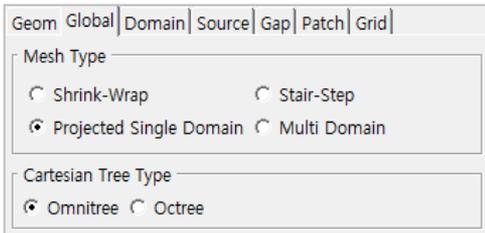
◆ Cartesian & Tetrahedral HYBRID 격자 생성

실행 방법

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



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



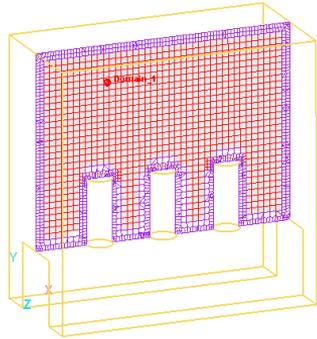
◆ Cartesian & Tetrahedral HYBRID 격자 생성

실행 방법

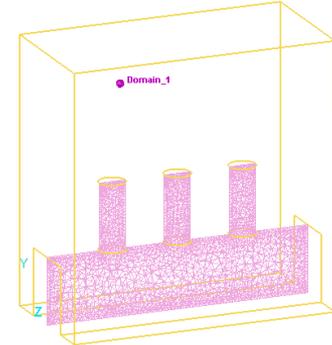
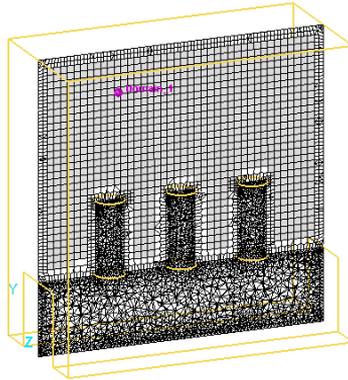
5. Generate Mesh 실행 후 Cutting Plan을 통해 확인

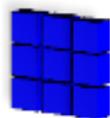


6. Domain 탭에서 각 항목을 선택하여 Domain 영역별 격자 확인 가능



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

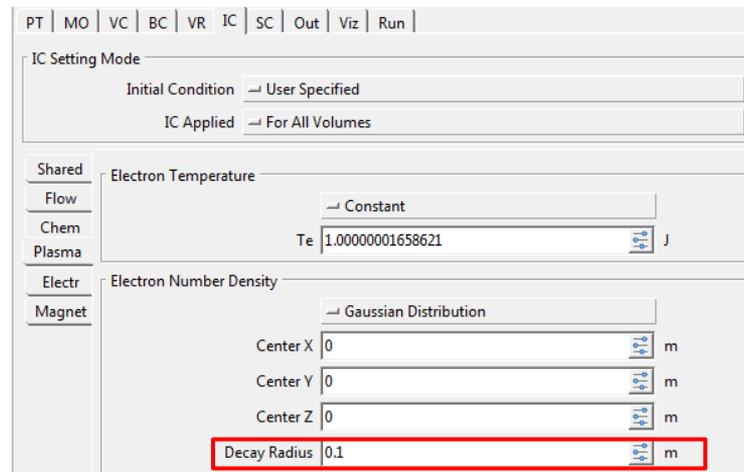
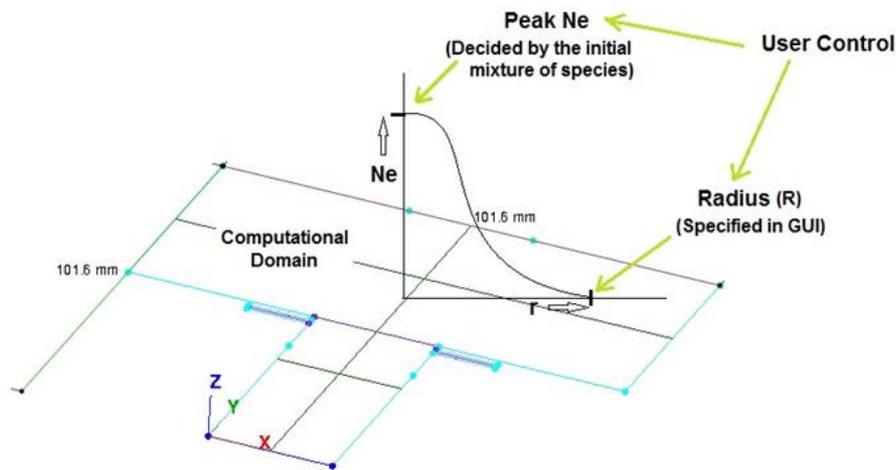




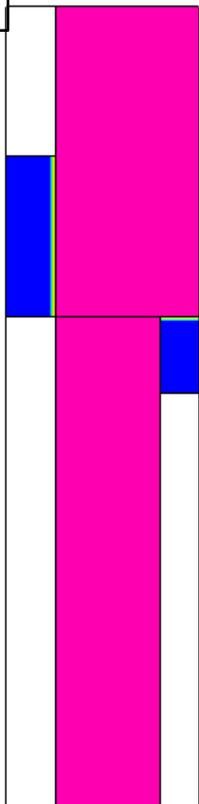
CFD-ACE+ V2020.0

◆ 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 줄어들며 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



V2019.0



IC Setting Mode

Initial Condition

IC Applied

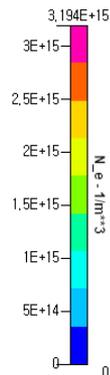
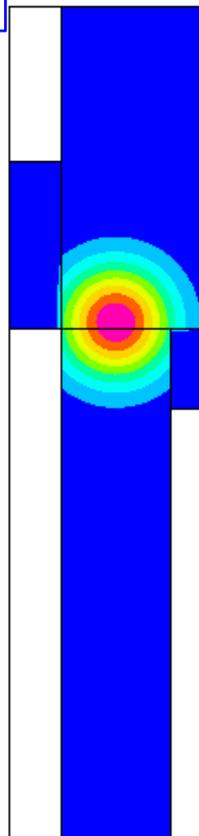
Shared

Flow

Chem
Plasma eV

Electr

V2020.0



IC Setting Mode

Initial Condition

IC Applied

Shared

Flow

Chem
Plasma eV

Electr

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:

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

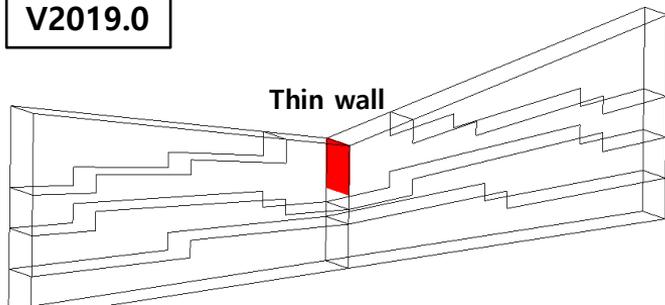
and

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

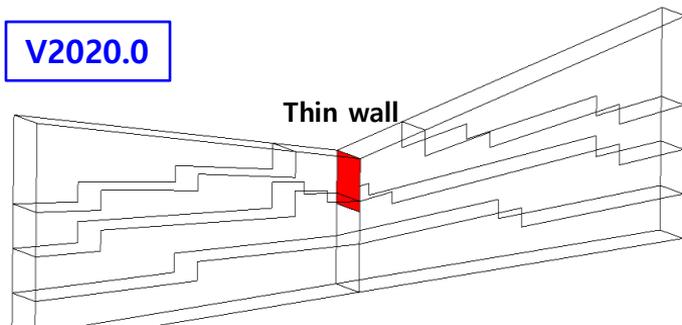
◆ Split interface boundary condition 병렬 처리 가능

- 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

V2019.0

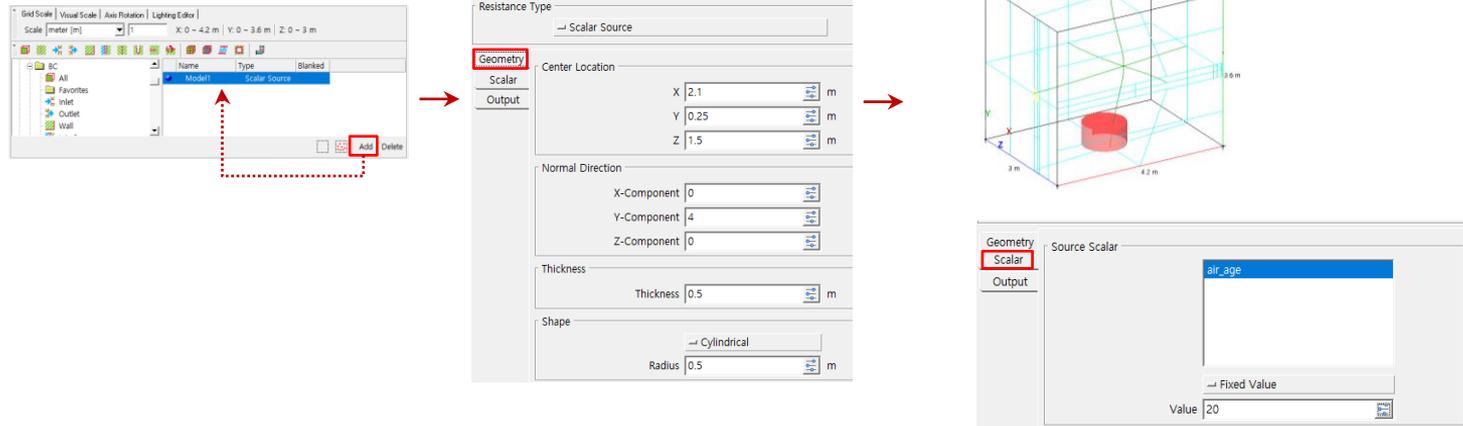


V2020.0



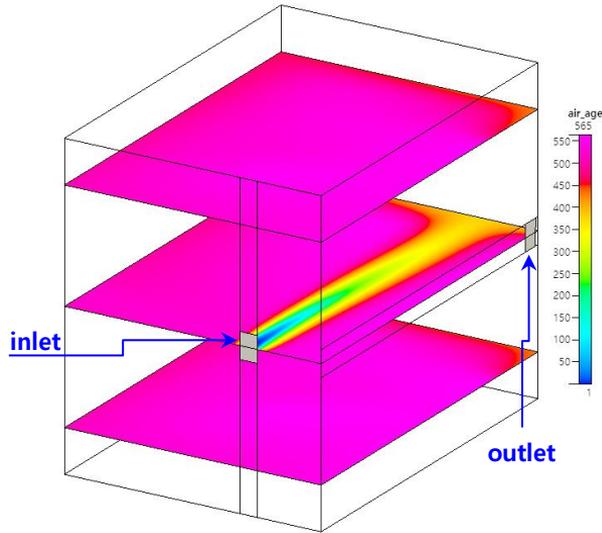
Virtual Resistance 탭에서 Scalar source 적용 가능

- 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

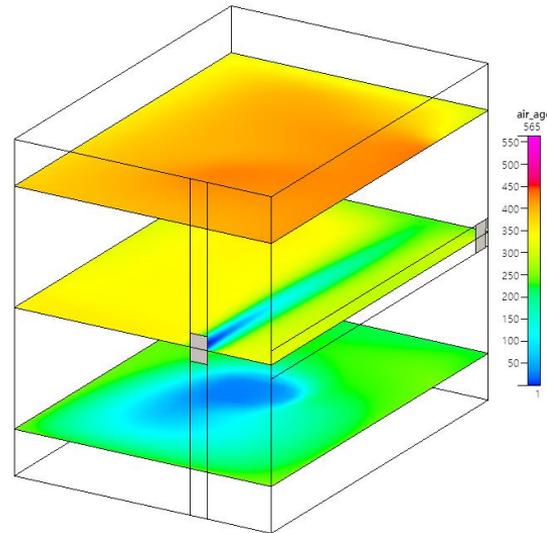


Virtual Resistance 탭에서 Scalar source 적용 가능

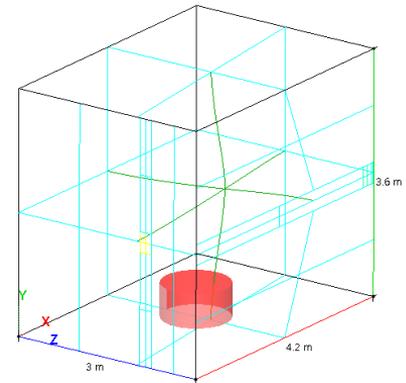
- Volume condition으로 지정하지 않고 사용자가 형상 지정 가능
 - Cylindrical, Polyhedral 선택 가능



Scalar source 적용 전

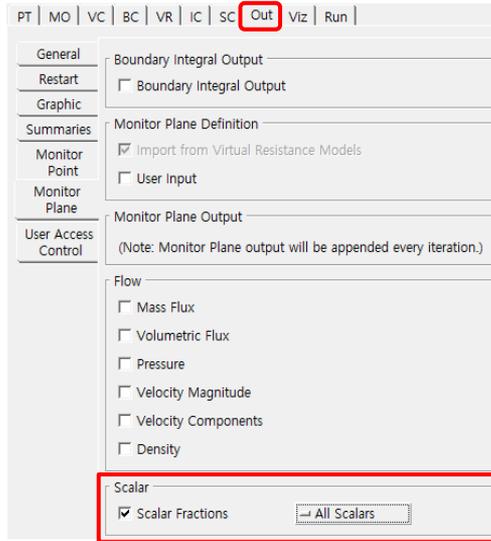


Scalar source 적용 후

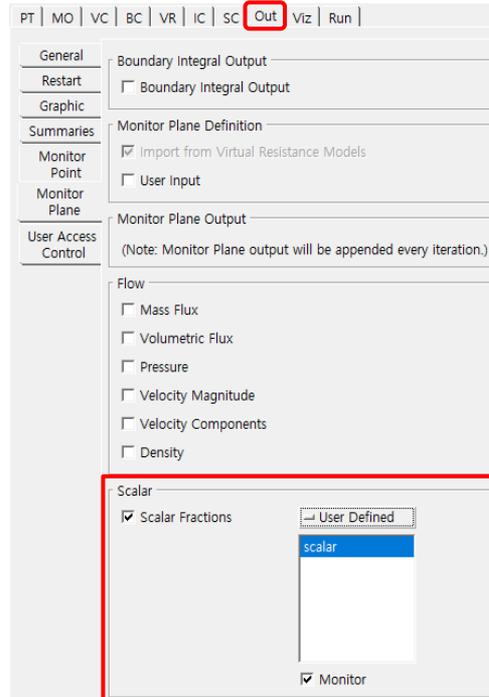


◆ 사용자가 지정한 평면에서 User Scalar 값 monitoring 가능

- 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.

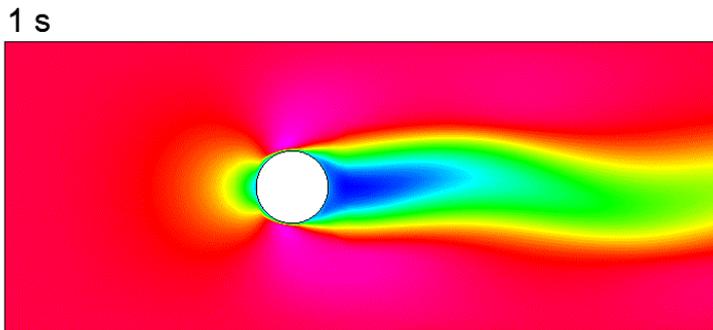
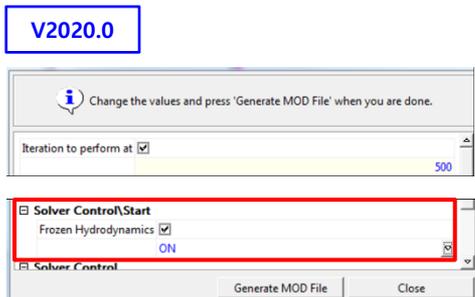
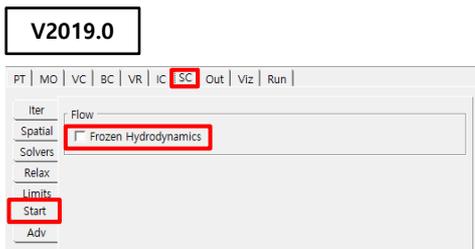
Solver Control\Numeric Solver Commands

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

◆ 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