비선형 동해석

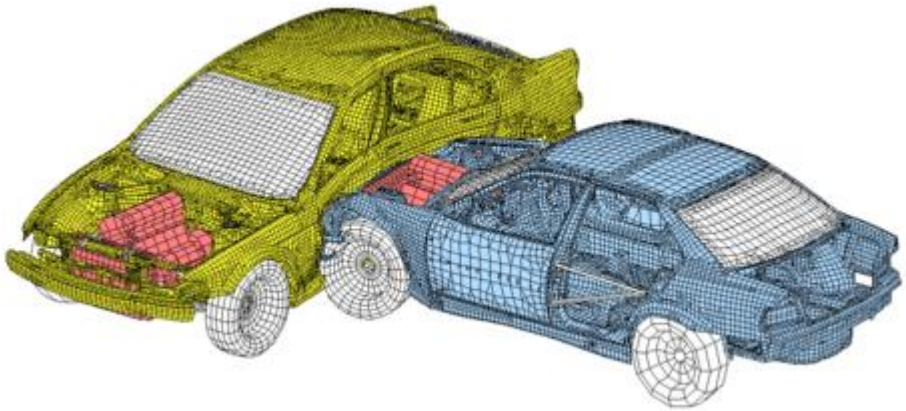
Computational Design Laboratory
Department of Automotive Engineering
Hanyang University, Seoul, Korea

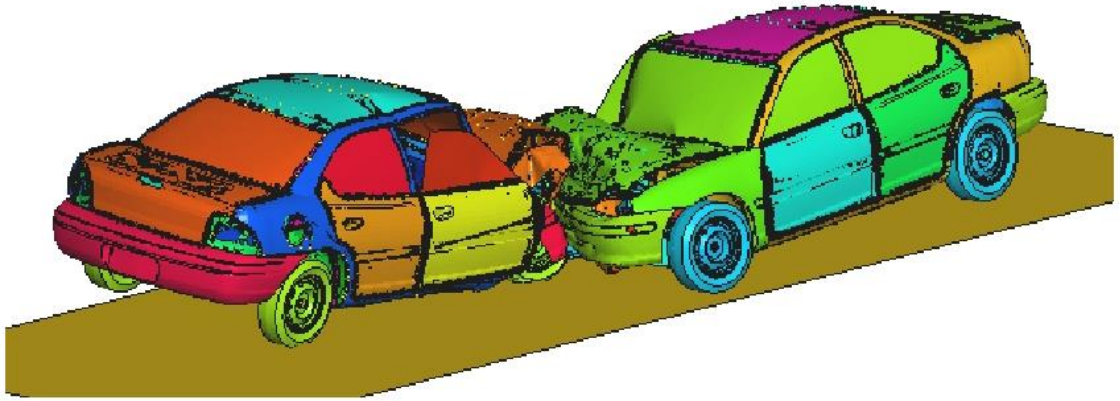
목차

- 예제 문제
  - Crash analysis
  - Beam crush analysis
- 해석 프로세스
  - 기하형상 생성
  - 재료 물성 및 특성 입력
  - 요소망 생성
  - 구속조건 설정
  - 하중조건 설정
  - 해석케이스 정의 및 해석 실행
  - 후처리

INTRODUCTION (1)

Crash analysis of body structure





INTRODUCTION (2)

표 7.7.1 암시적(Implicit)과 외연적(Explicit) 적분 알고리즘 비교

암시적(Implicit) 적분 알고리즘	외연적(Explicit) 적분 알고리즘
• 선형 과도응답해석의 경우 큰 시간 스텝을 적용할 수 있음 • 비선형 과도응답해석의 경우 수렴성 확보를 위해 작은 시간 스텝을 사용해야 하는 단점이 있음 • 대규모 모델인 경우에는 메모리 사용량 및 하드디스크 용량의 제한이 있을 수 있음	• 수렴성에 대한 문제 없이 해가 구해짐 • 소규모 모델인 경우에는 암시적 방법에 비해 해석 시간이 좀 더 많이 소요되나 대규모 모델에서는 상대적으로 유리함 • 시간 간격은 항상 임계 시간 간격보다 작은 값으로 정의

$$f(x_{n+1}, x_n, \dots) = 0$$

$$x_{n+1} = f(x_n, \dots)$$

INTRODUCTION (3)

- 내연적 시간 적분법 (HHT-α 이용: Newmark 방법의 일반화된 형태)

1. 동적 평형방정식으로부터 구한 불평형력을 최소화 하는 방향으로 진행

동적 평형방정식: $\mathbf{Ma}^{n+1}+(1+\alpha_H)\left[\mathbf{Cv}^{n+1}+\mathbf{f}^{\text{int},n+1}-\mathbf{f}^{ext,n+1}\right]-\alpha_H\left[\mathbf{Cv}^{n}+\mathbf{f}^{\text{int},n}-\mathbf{f}^{ext,n}\right]=\mathbf{0}$

불평형력: $\mathbf{g}_{n+1}=\mathbf{M}\ddot{\mathbf{u}}_{n+1}+(1+\alpha_H)\left(\mathbf{C}_{n+1}\dot{\mathbf{u}}_{n+1}+\mathbf{f}_{\text{int},n+1}-\mathbf{f}_{ext,n+1}\right)-\alpha_H\left(\mathbf{C}_{n}\dot{\mathbf{u}}_{n}+\mathbf{f}_{\text{int},n}-\mathbf{f}_{ext,n}\right)$

2. 변위 및 가속도 계산

$$\mathbf{v}^{n+1}=\mathbf{v}^{n}+\Delta t\left[\gamma\mathbf{a}^{n+1}+(1-\gamma)\mathbf{a}^{n}\right]$$

$$\mathbf{u}^{n+1}=\mathbf{u}^{n}+\Delta t\mathbf{v}^{n}+\frac{1}{2}\Delta t^{2}\left[2\beta\mathbf{a}^{n+1}+(1-2\beta)\mathbf{a}^{n}\right]$$

3. 평형방정식 재구성

$$\mathbf{K}^{eff}\mathbf{u}_{n+1}=\mathbf{f}^{eff}$$

$$\mathbf{K}^{eff}=\frac{1}{\beta\Delta t^{2}}\mathbf{M}+\frac{(1+\alpha_H)\gamma}{\beta\Delta t}\mathbf{C}+(1+\alpha_H)\mathbf{K},$$

$$\mathbf{f}^{eff}=-\mathbf{f}^{\text{int},0}+(1+\alpha_H)\left[\mathbf{f}^{ext,n+1}+\mathbf{f}^{nonmech,n+1}\right]-\alpha_H\left[\mathbf{f}^{ext,n}+\mathbf{f}^{nonmech,n}\right]+$$

$$\mathbf{M}\left[\frac{1}{\beta\Delta t^{2}}\mathbf{u}^{n}+\frac{1}{\beta\Delta t}\mathbf{v}^{n}+\left(\frac{1}{2\beta}-1\right)\mathbf{a}^{n}\right]+$$

$$\mathbf{C}\left[\frac{(1+\alpha_H)\gamma}{\beta\Delta t}\mathbf{u}^{n}+\left\{\frac{(1+\alpha_H)\gamma}{\beta}-1\right\}\mathbf{v}^{n}+\Delta t(1+\alpha_H)\left(\frac{\gamma}{2\beta}-1\right)\mathbf{a}^{n}\right]+\alpha_H\mathbf{K}\mathbf{u}^{n}$$

INTRODUCTION (4)

- 외연적 시간 적분법 (중앙차분법 이용)

1. 시간 스텝을 n+1/2과 n, n+1 스텝으로 구분

$$\Delta t^{n+1/2} = t^{n+1} - t^n,\ t^{n+1/2} = \frac{1}{2}\left(t^{n+1} + t^n\right),\ \Delta t^n = t^{n+1/2} - t^{n-1/2}$$

2. 스텝 n+1 에서의 변위는 n+1/2 스텝에서의 속도로부터 계산

$$\dot{\mathbf{u}}^{n+1/2} = \mathbf{v}^{n+1/2} = \frac{1}{\Delta t^{n+1/2}}\left(\mathbf{u}^{n+1} - \mathbf{u}^n\right),\quad \mathbf{u}^{n+1} = \mathbf{u}^n + \Delta t^{n+1/2}\mathbf{v}^{n+1/2}$$

3. 스텝 n+1/2 에서의 속도는 n 스텝에서의 가속도로부터 계산

$$\ddot{\mathbf{u}}^n = \mathbf{a}^n = \frac{1}{\Delta t^n}\left(\mathbf{v}^{n+1/2} - \mathbf{v}^{n-1/2}\right),\quad \mathbf{v}^{n+1/2} = \mathbf{v}^{n-1/2} + \Delta t^n \mathbf{a}^n$$

4. n 스텝의 가속도 a 는 공간상의 이산화를 통해 계산

$$\mathbf{M}\mathbf{a}^n = \mathbf{f}^n = \mathbf{f}^{ext}(\mathbf{u}^n, t^n) - \mathbf{f}^{\text{int}}(\mathbf{u}^n, t^n),\ \mathbf{a}^n = \mathbf{M}^{-1}\left(\mathbf{f}^{ext}(\mathbf{u}^n, t^n) - \mathbf{f}^{\text{int}}(\mathbf{u}^n, t^n)\right)$$

5. 1~4과정 반복

- 외연적 시간 적분법의 임계 시간스텝

해석 모델에 포함된 모든 요소의 안정시간스텝 중 가장 작은 값을 기준으로 계산

$$\Delta t = \alpha \Delta t_{crit},\ \Delta t_{crit} = \frac{2}{\omega_{\max}} \le \min_e\left\{\Delta t_e\right\} \qquad \Delta t_e = \frac{2}{\omega^e_{\max}} = \min\left\{\frac{L_e}{c_d}\right\}$$

$\omega_{\max}$ 는 선형화된 시스템 전체의 최대 주파수

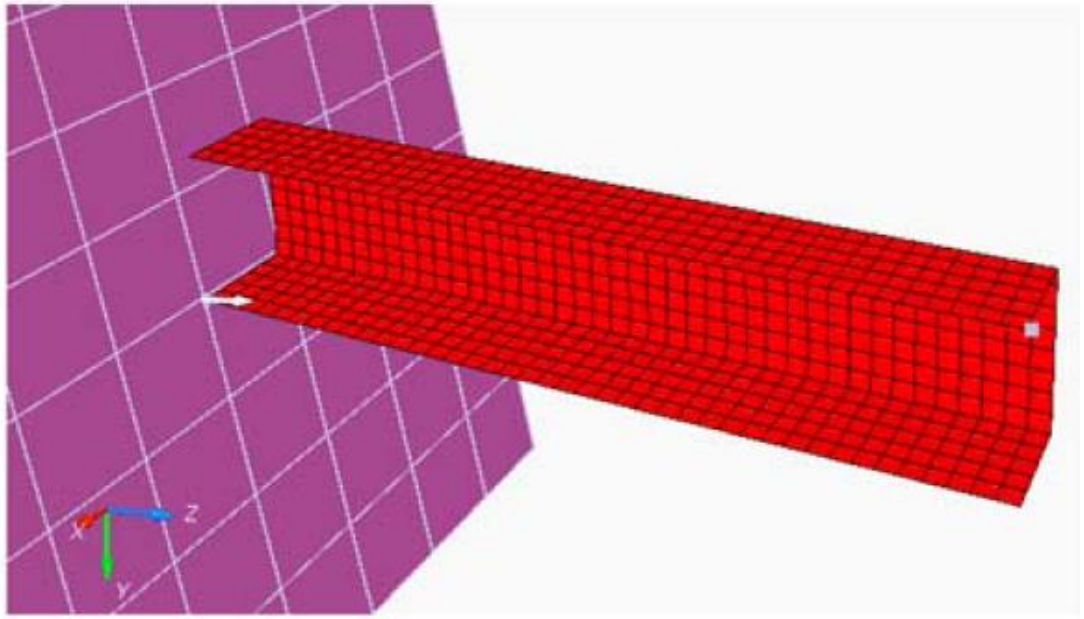
L_e 는 요소의 특성길이

c_d 는 팽창파 속도 $\quad c_d = \sqrt{\frac{M}{\rho}} = \sqrt{\frac{K + 4\mu/3}{\rho}}$

예제: BEAM CRASH ANALYSIS

Simulate buckling of a tube using half tube mesh with symmetric boundary conditions.

The figure illustrates the structural model used for this tutorial: a half tube with a rectangular section (38.1 x 25.4 mm) and length of 203 mm.



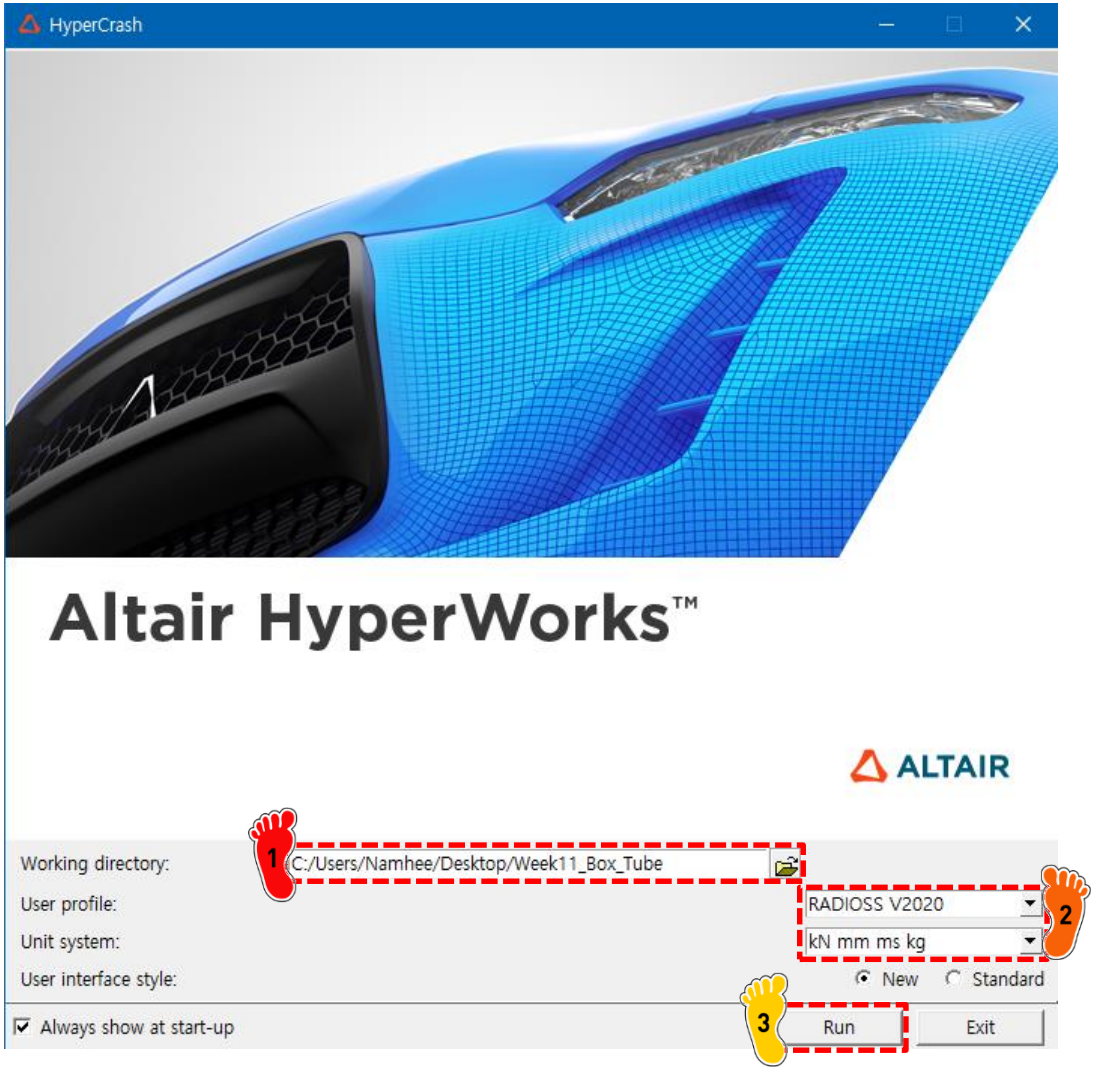
Model

- The tube thickness is 0.914 mm.

- $\rho = 7.85e^{-6}$ Kg/mm^3 — Initial density
- E = 210 GPa — Young's modulus
- v = 0.33 — Poisson coefficient
- σ_0 = 0.206 GPa — [a] Yield Stress

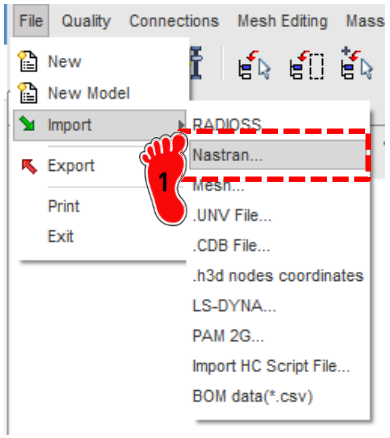
BEAM CRASH EXAMPLE

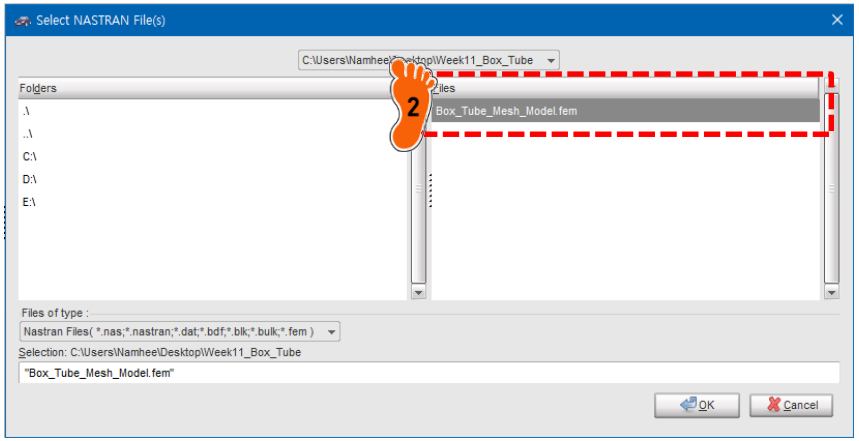
HYPERCRASH 실행

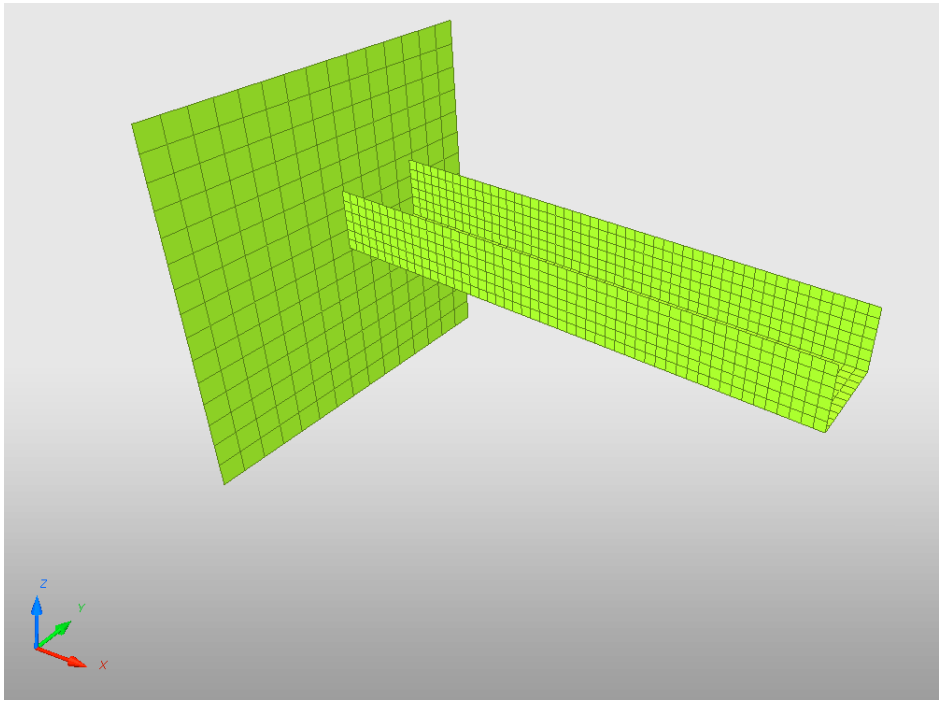


1. HyperCrash2020 실행 후 Working directory 설정 (모델 파일이 있는 폴더로 지정)
2. User profile, Unit system 설정
3. Run 클릭

유한요소해석 모델 불러오기



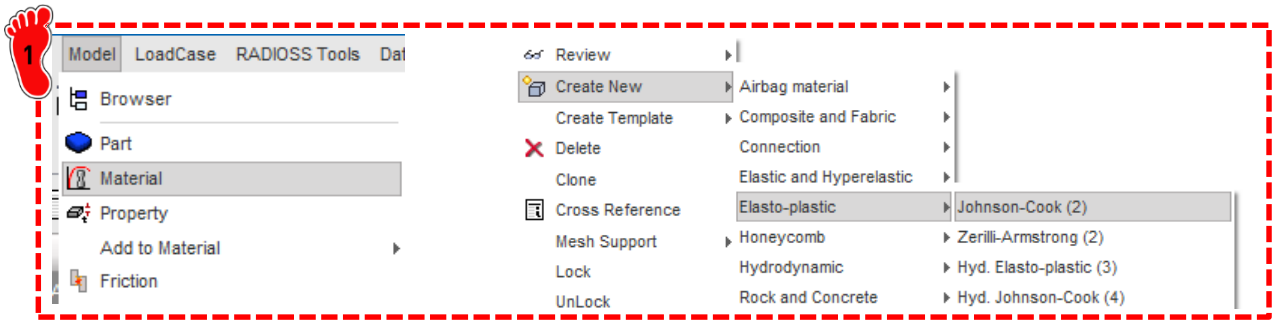


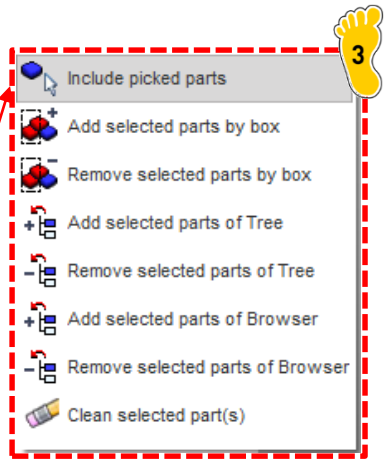


1 File > Import > Nastran 클릭

2 모델 파일 클릭 후 불러오기 (Box_Tube_Mesh_Model.fem)

재료 물성 입력



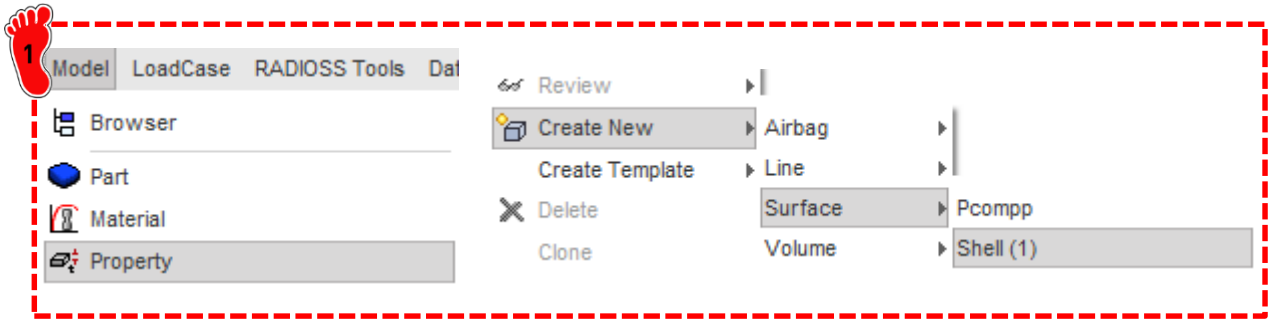


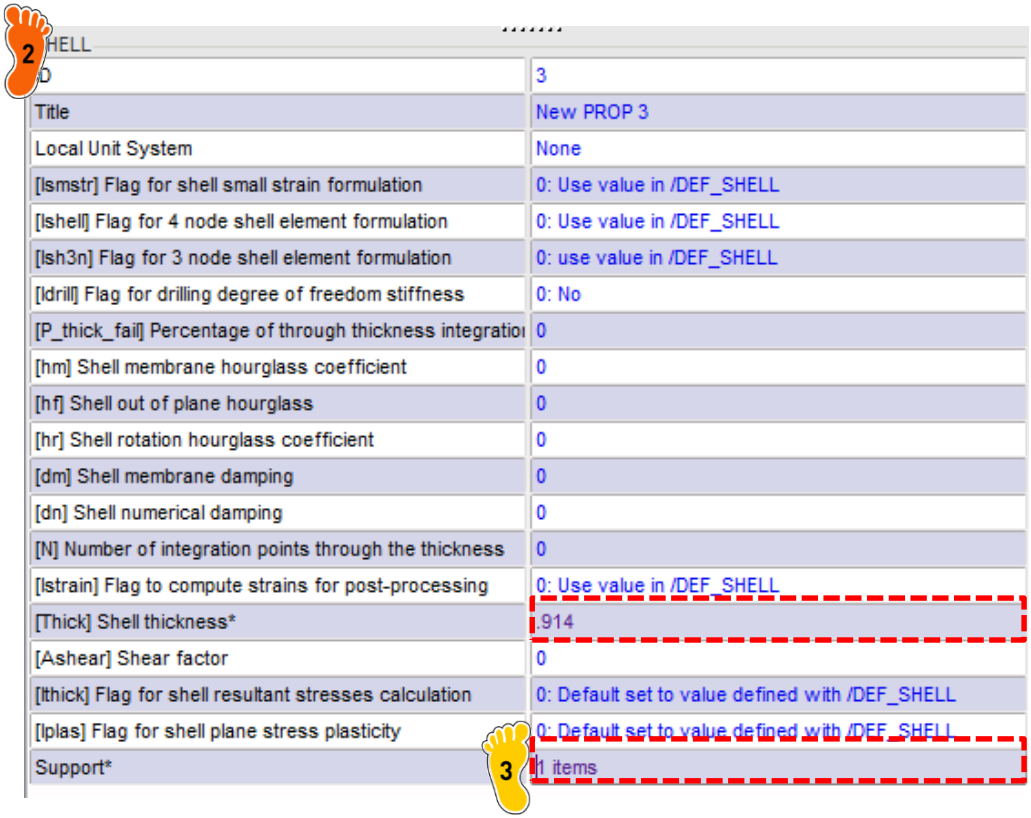
1 Model > Material 클릭, 마우스 우클릭 후 Create New > Elasto-plastic > Johnson-Cook (2) 클릭

2 재료 물성치 입력
[RHO_I]: 7.85e-6
[E]: 210
[Nu]: 0.3
[a]: 0.206

3 Support* 항목의 경우 마우스 우클릭하여 "Select in graphics" 선택, Include picked part ( ) 클릭 후 모델 선택

특성 입력



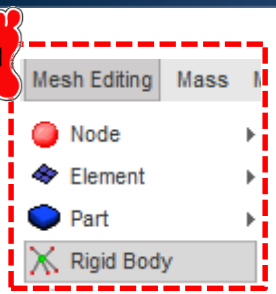


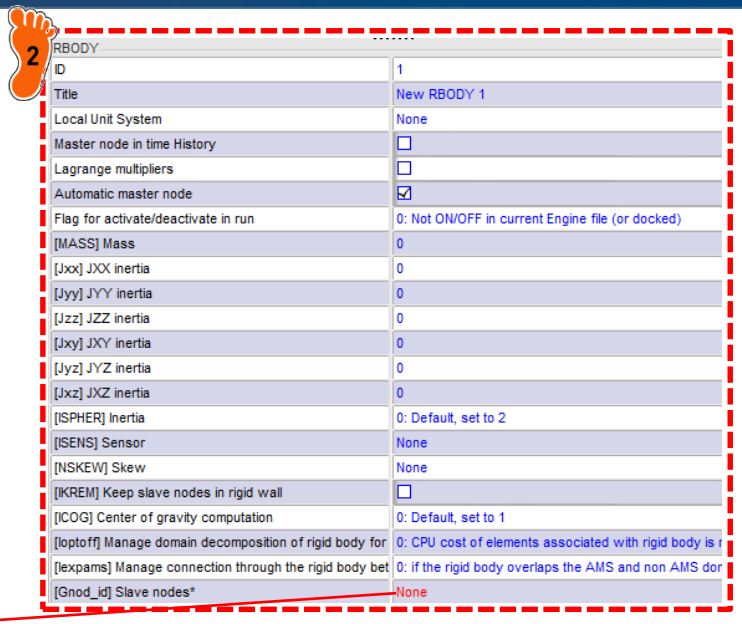
1 Model > Material 클릭, 마우스 우클릭 후 Create New > Elasto-plastic > Johnson-Cook (2) 클릭

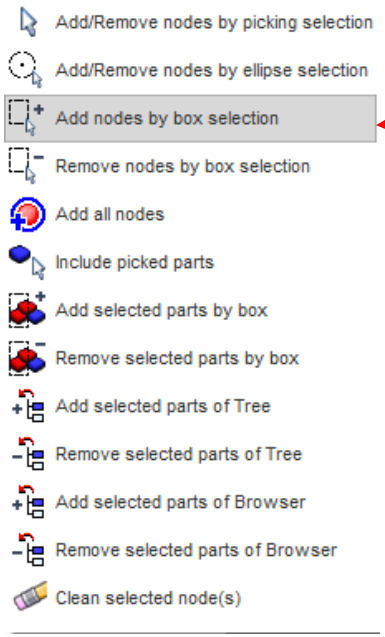
2 재료 물성치 입력
[Thick]: 0.914

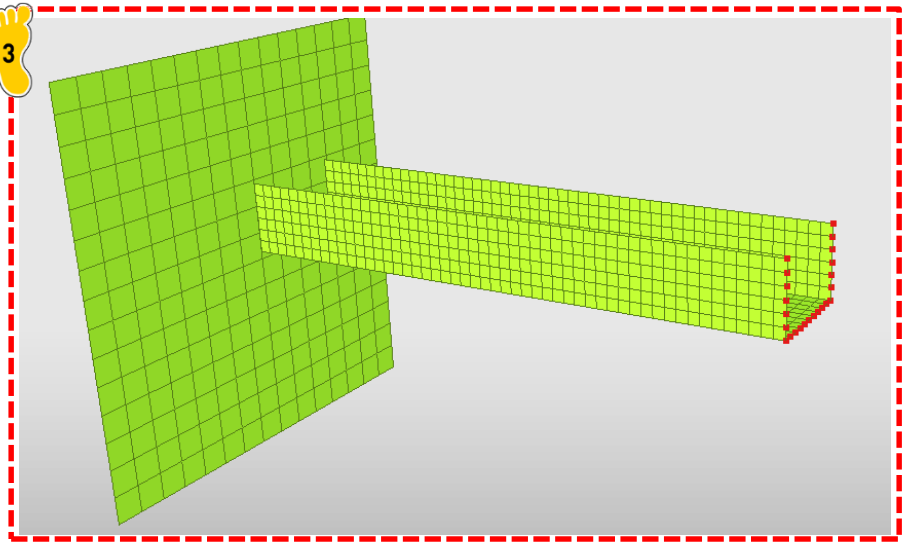
3 Support* 항목의 경우 마우스 우클릭하여 "Select in graphics" 선택, Include picked part ( ) 클릭 후 모델 선택

강체 요소 생성







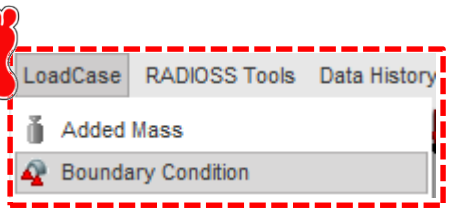


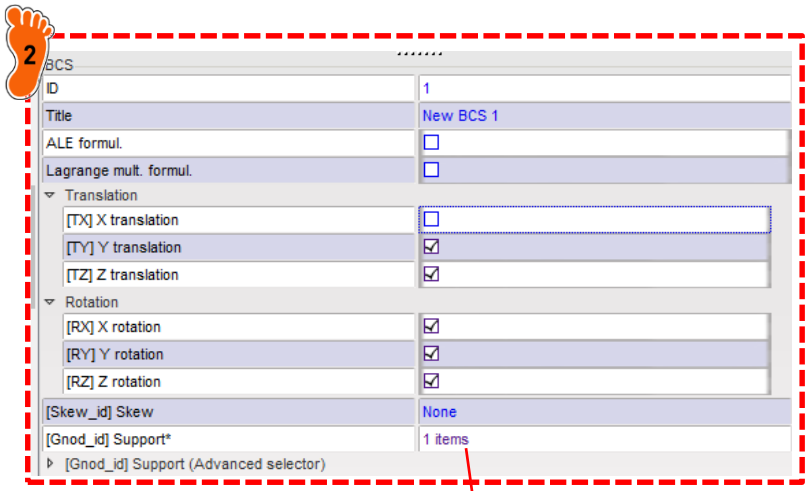
1 Mesh Editing > Rigid Body 클릭, 마우스 우클릭 후 Create New 클릭

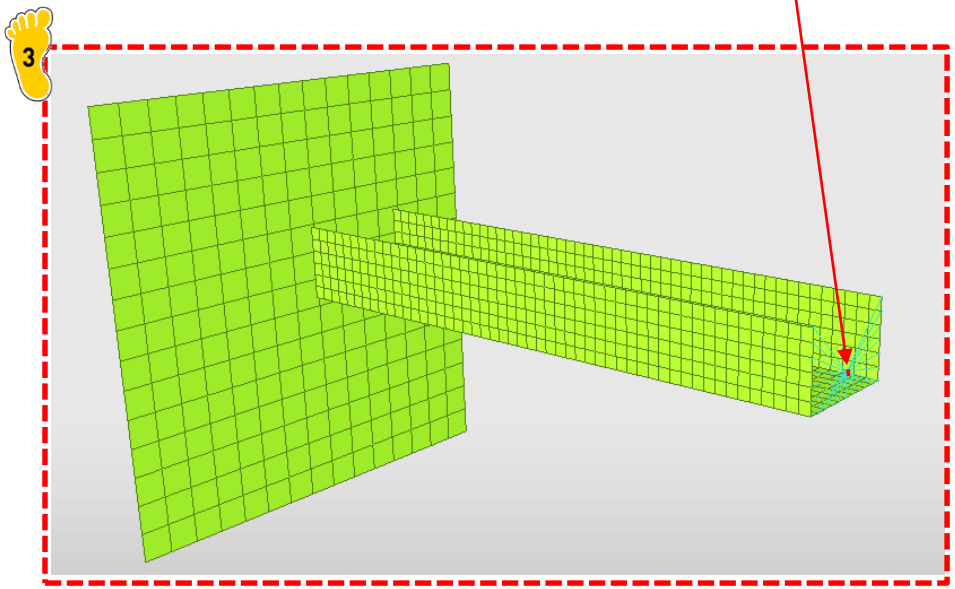
2 Slave nodes 항목에서 마우스 우클릭 후 Select in graphics 선택 > Add nodes by box selection 선택

3 그림과 같이 tube 끝 절점 선택 후 save 클릭

구속조건 설정 (1)





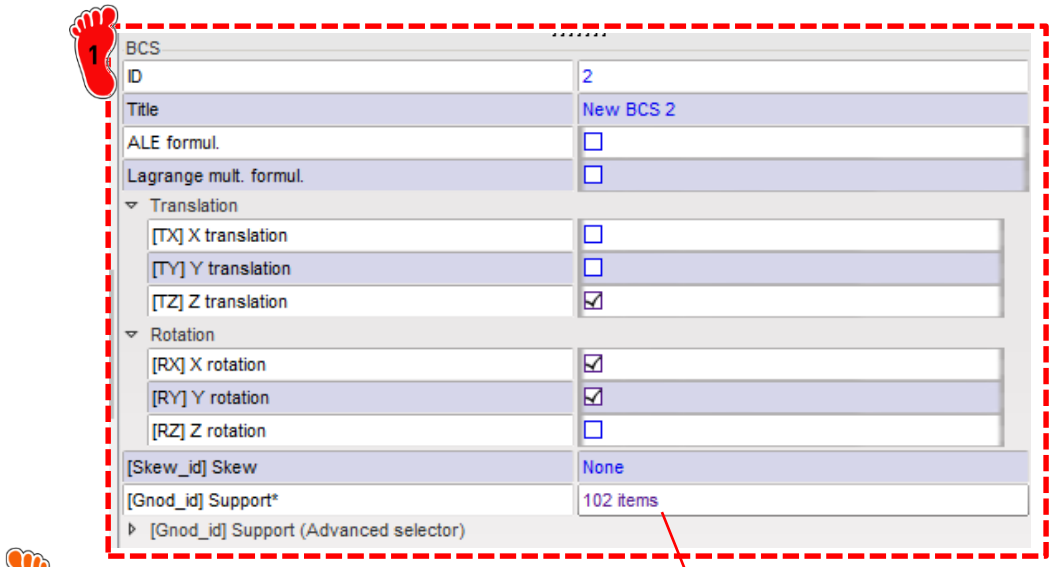


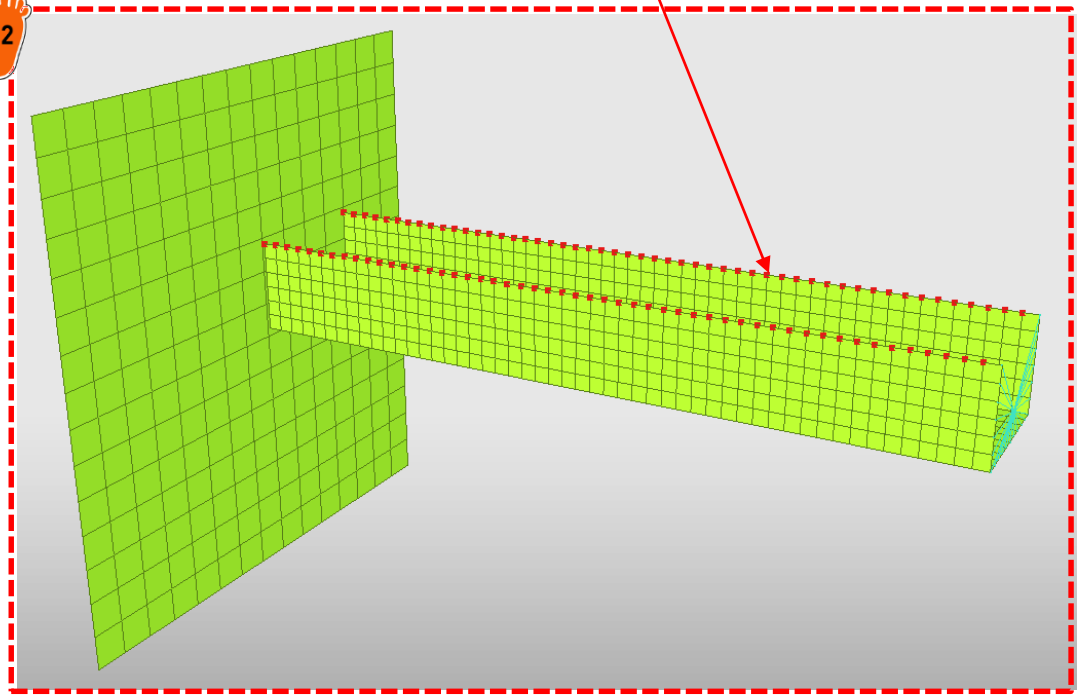
1 LoadCase > Boundary Condition 클릭, 마우스 우클릭 후 Create new 클릭

2 TX 제외하고 모두 체크

3 앞서 제작한 Rigid 요소의 Master node 선택 후 save 클릭

구속조건 설정 (2)

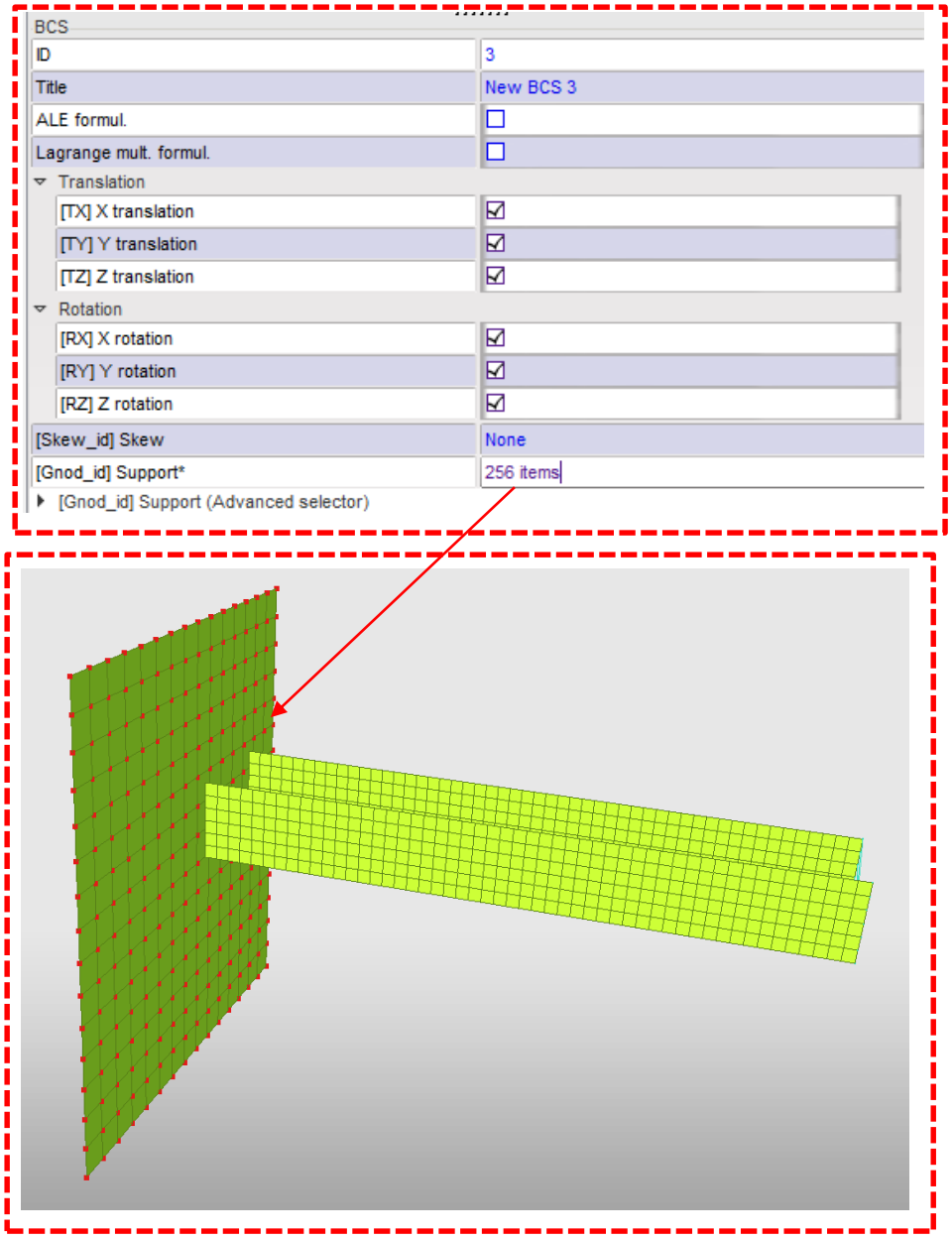




1 경계조건 추가 후 RZ, RX, RY 체크

2 그림과 같이 tube의 상단 절점 선택 후 save 클릭

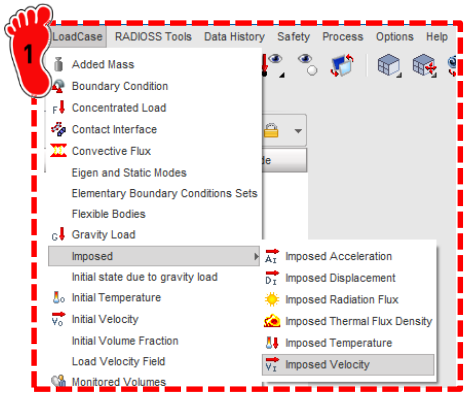
구속조건 설정 (3)

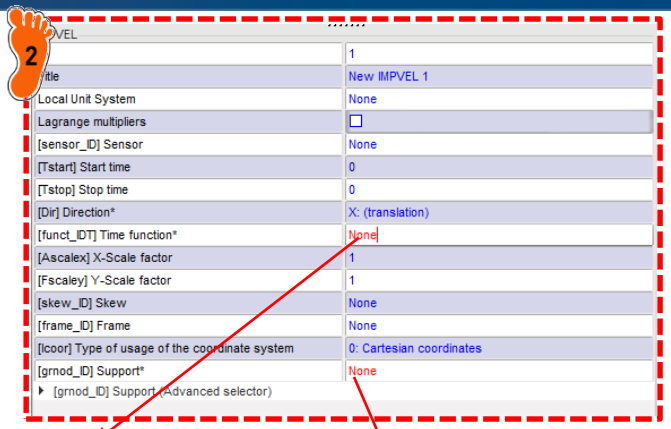


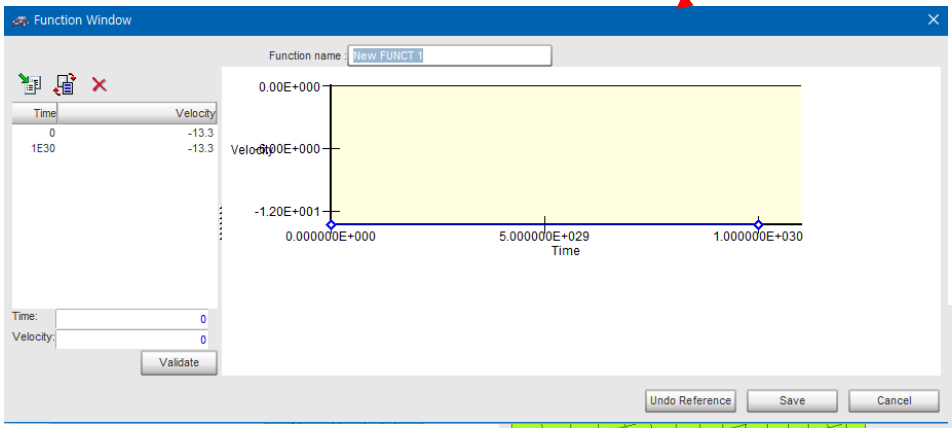
1 경계조건 추가 후 자유도 모두 체크

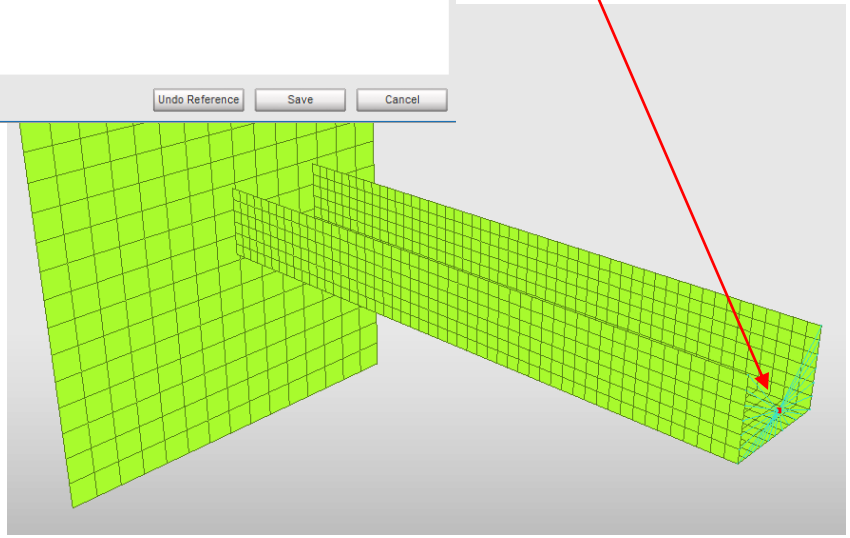
2 그림과 같이 벽면 절점 선택 후 save 클릭

TUBE 속도 설정







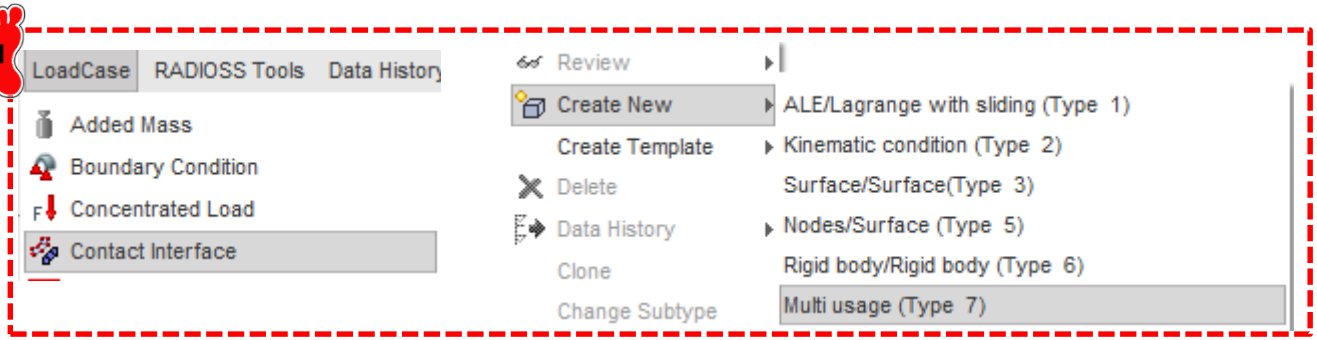


1 LoadCase > Imposed > Imposed Velocity 클릭, 마우스 우클릭 후 Imposed velocity 생성

2 Time function 생성 (Time, Velocity)
→ (0, -13.3) ~ (1e30, -13.3)

Support → Rigid 요소의 마스터 절점 선택

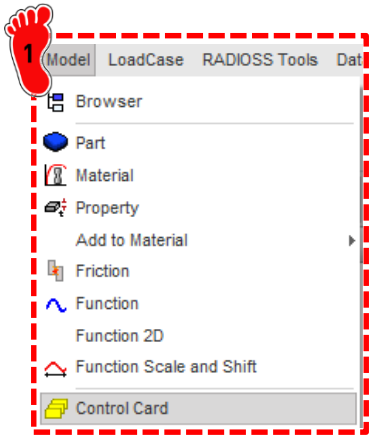
접촉조건 설정

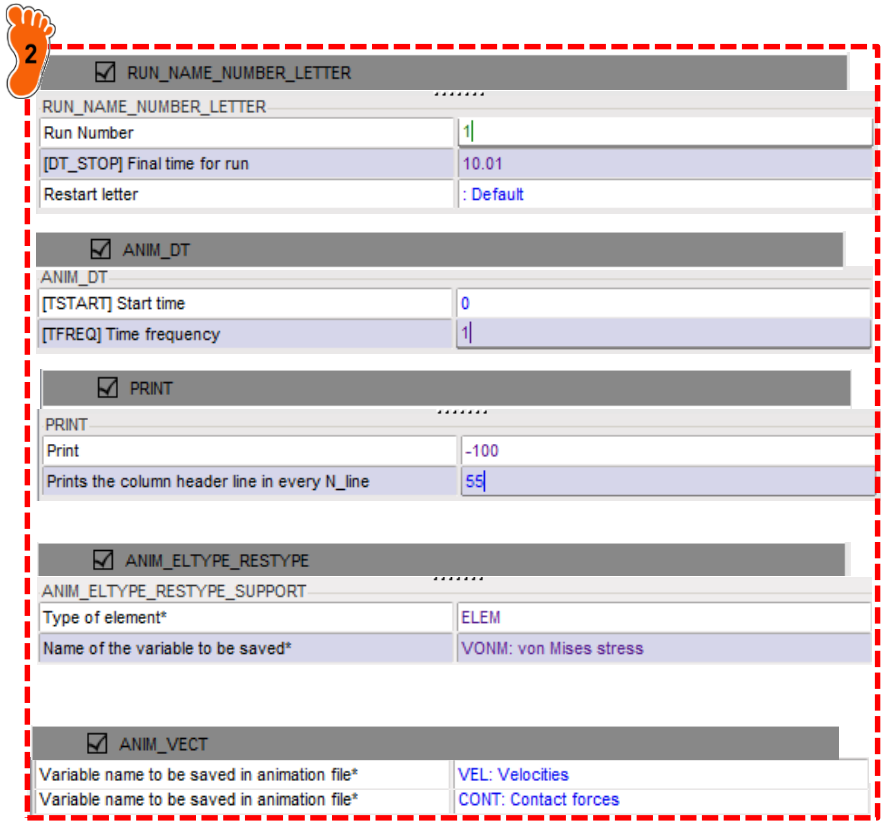


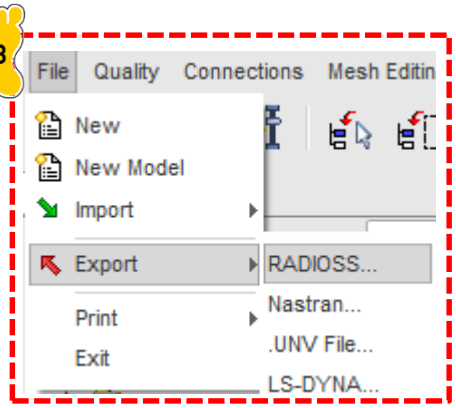
1 LoadCase > Contact Interface클릭,
마우스 우클릭 후 "Multi usage (Type 7) 클릭

2 Self Impact 체크
[Stfac] : 1
[Gapmin]: 0.9
[Fric]: 0.2
[Mast_id]: 모델 선택

해석 케이스 설정







1 Model > Control Card클릭

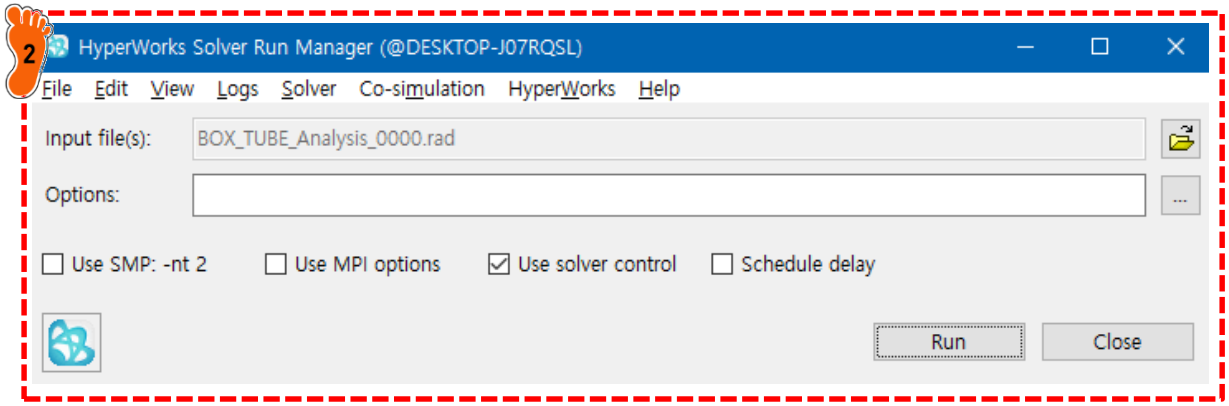
2 파라미터 설정

3 모델 export (RADIOSS 선택)

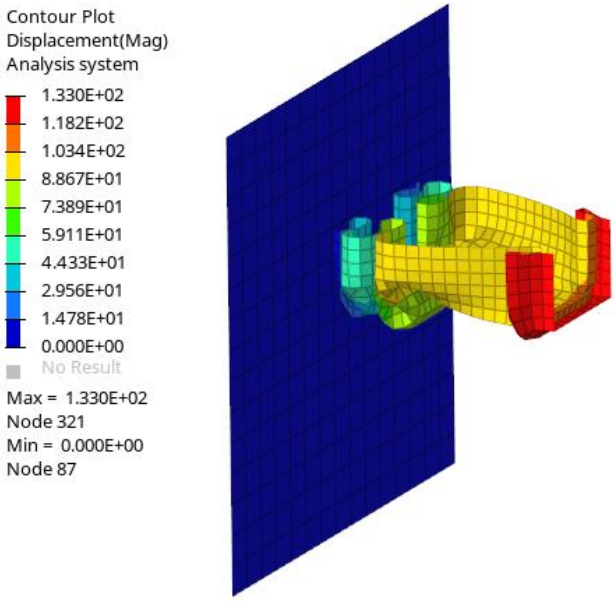
해석 실행 (RADIOSS)

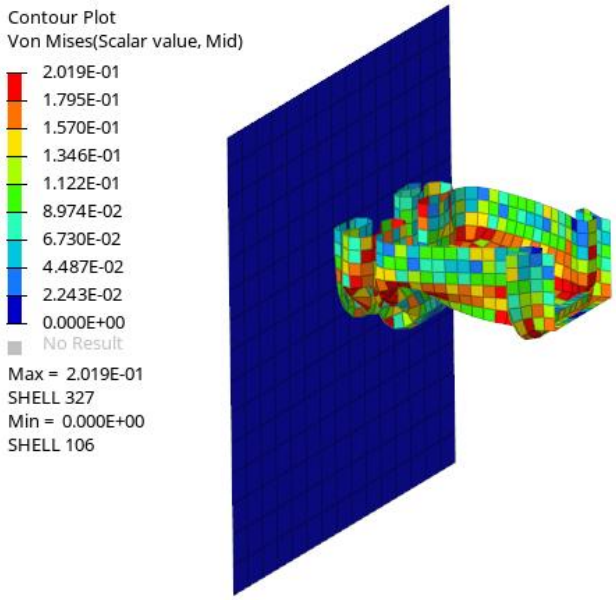
1 Radioss2020 실행

2 앞서 제작한 모델 불러온 뒤 Run 클릭



후처리





1 변위, 응력 등 결과 확인

연습문제: IMPACT OF A ROD ON RIGID WALL

Figure 12.6.1 shows a cylindrical rod model to simulate a high velocity impact event in which the cylindrical rod collides with a rigid wall. The collision is modeled by imposing zero axial displacement prescribed at one end of the rod, while imposing an initial axial velocity of 8937 in/sec to all other nodes. A von Mises elastic-perfectly plastic material model with isotropic hardening is used. The length and radius at 80 micro-seconds after the impact are obtained and compared with the reference values. Nonlinear explicit transient analysis is performed and the initial mesh and deformed shapes at 40 and 80 micro-seconds after the impact are shown in Figure 12.6.2.

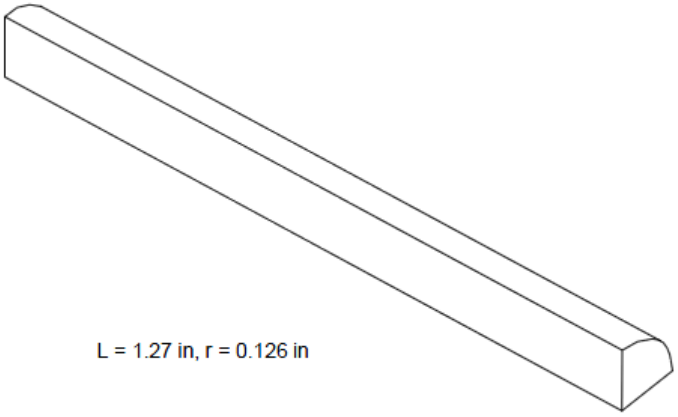


Figure 12.6.1 A rod impact model

Material data	Young's modulus	$E = 17\ msi$
	Poisson's ratio	$\nu = 0.35$
	Density	$\rho = 0.3224\ lbm/in^3$
	Hardening Modulus	$E_T = 14.5\ ksi$
	Yield Stress	$\sigma_Y = 58\ ksi$

Table 12.6.1 The deformed length and radius at t=80 μsec after impact

	Length [in]	Radius [in]
Reference	0.84	0.28
midas-NFX	0.84	0.26

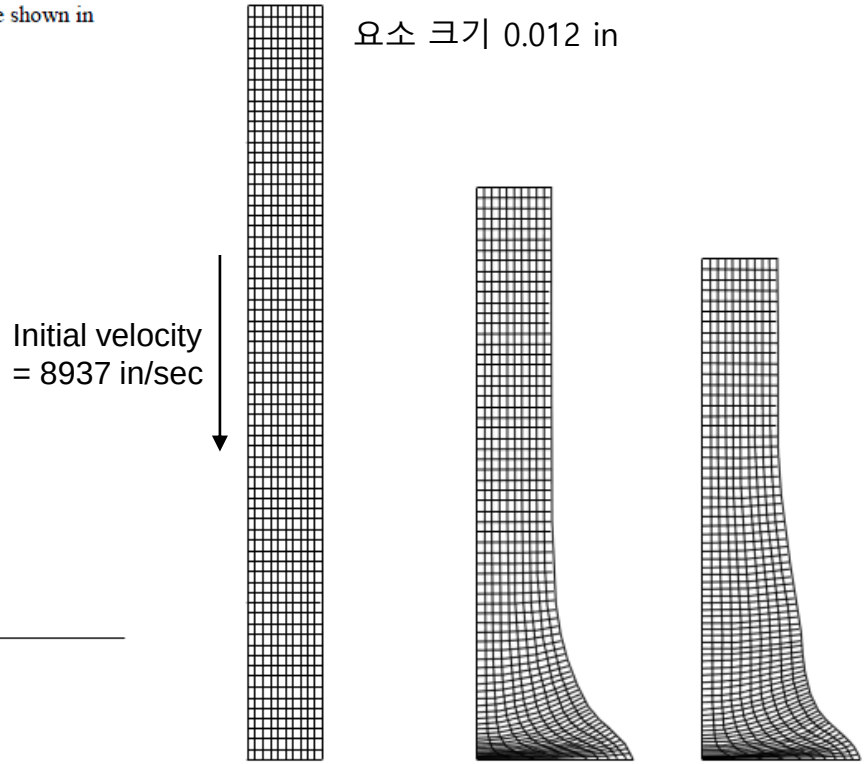


Figure 12.6.2 Deformed shape of the rod at t=0, 40 and 80 μsec

BEAM CRUSH ANALYSIS
쉘 요소

예제: BEAM CRUSH ANALYSIS (1)

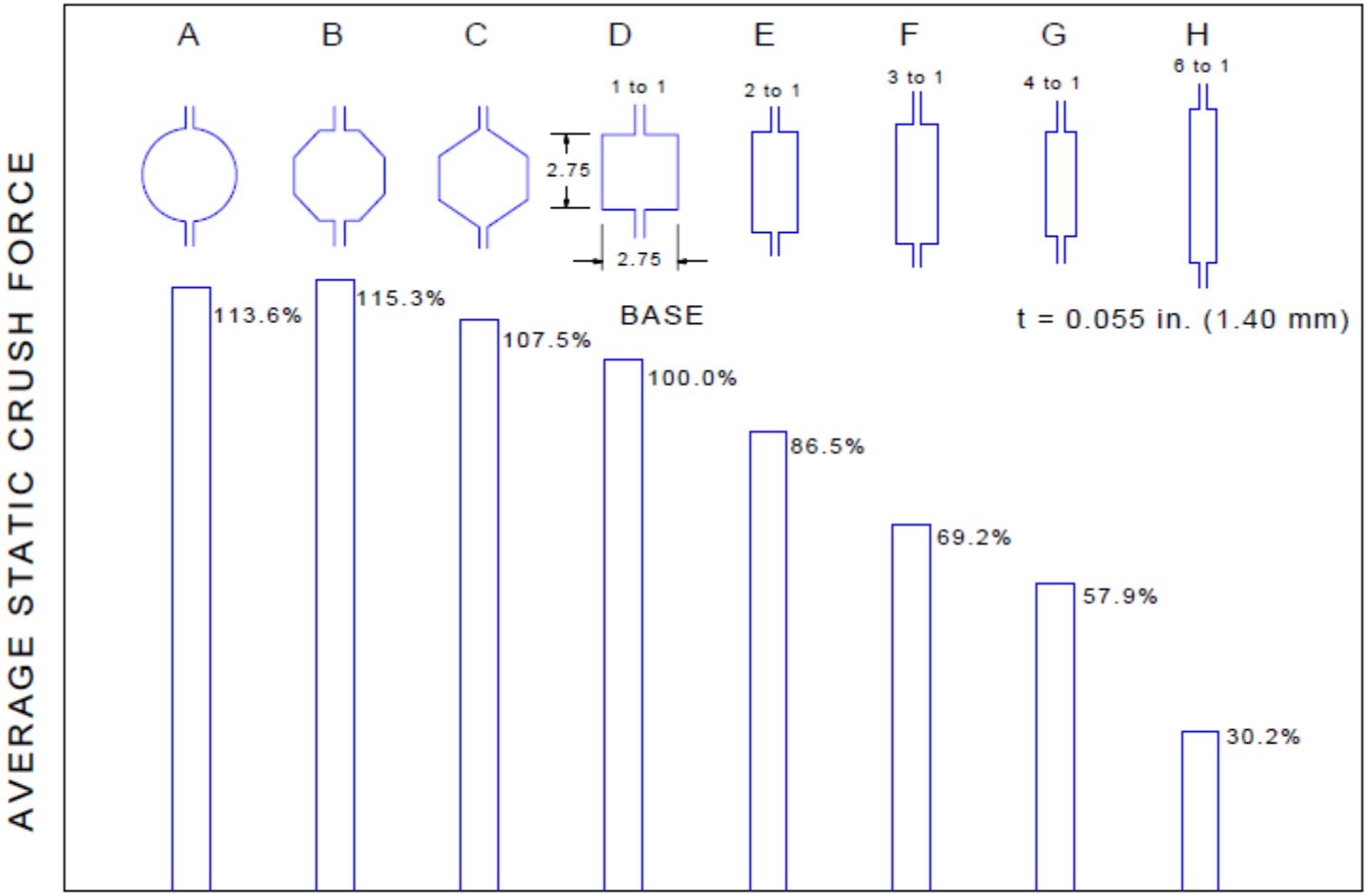
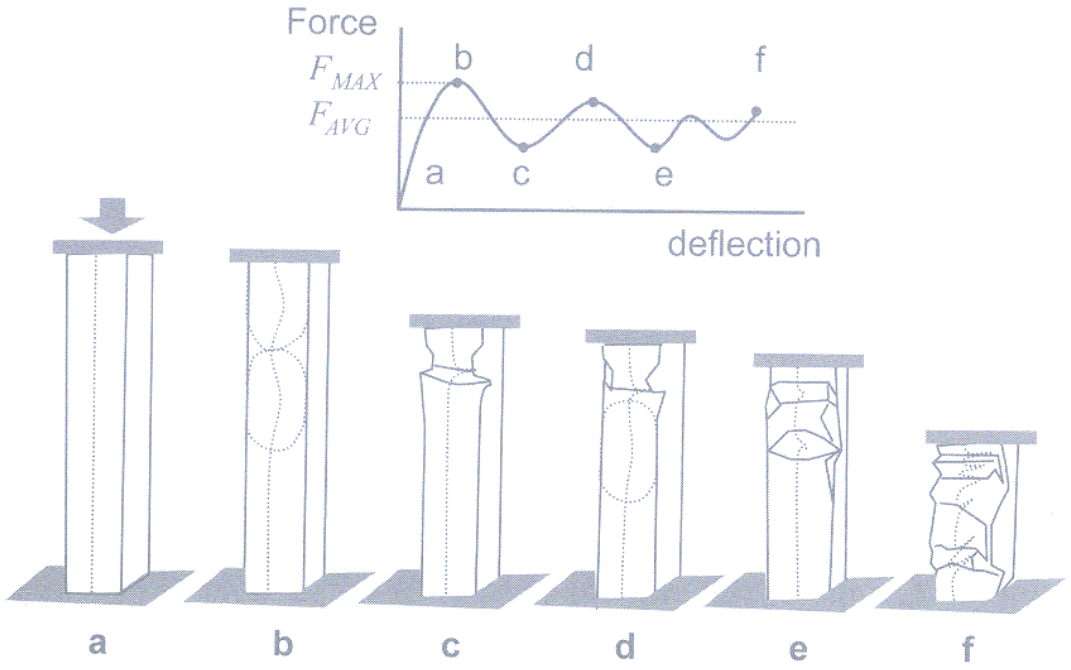


Figure 3.6.4-1 Average static crush force vs. section shape
(All samples were the same mass and length)

예제: BEAM CRUSH ANALYSIS (2)

Geometry information

Length: 305 mm
70 mm square
thickness 1.4 mm
Average strength 247 MPa



$$P_M = 386 t^{1.86} b^{0.14} \sigma_Y^{0.57}$$

$$P_{MAX} = 2.87 P_M$$

$$P_1 = 1.42 P_M$$

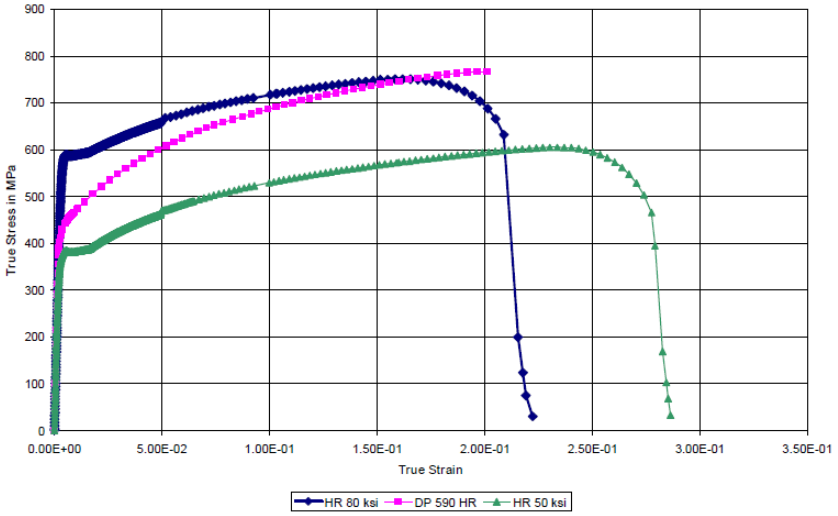
$$P_2 = 0.57 P_M$$

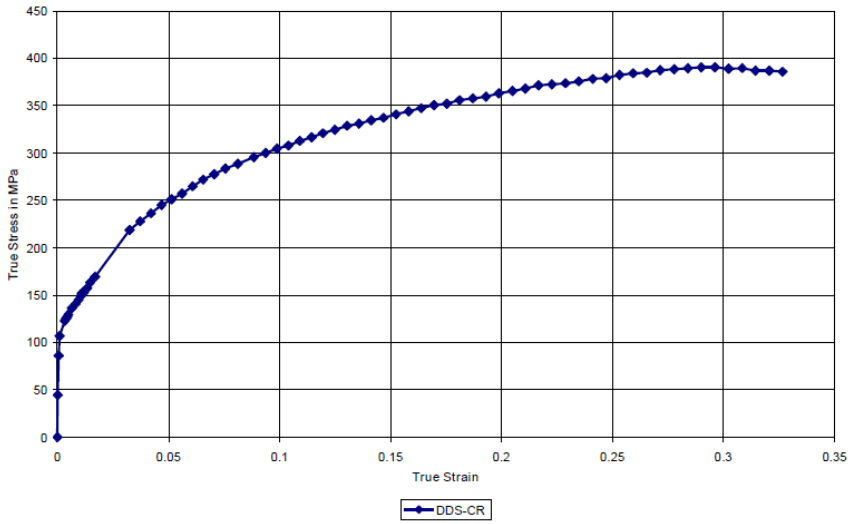
예제: 재료 특성

Some of the material properties required to predict the crush characteristics are:

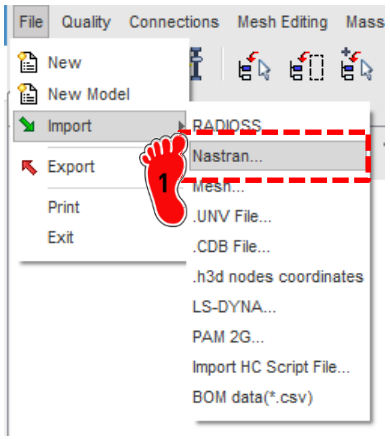
1. Stress-strain properties representative of the material for large plastic deformation
2. Change in properties under dynamic loading conditions
3. Strain hardening
4. Ductility properties
5. The variability that can be expected for production steels

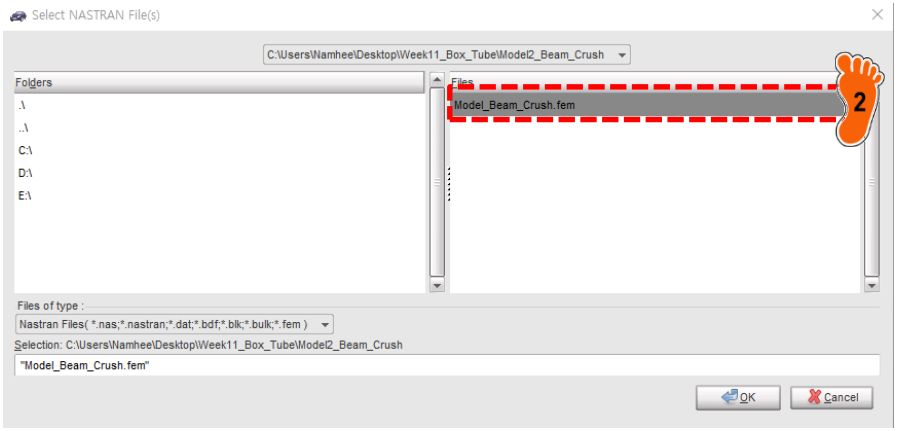
Stress-strain curves

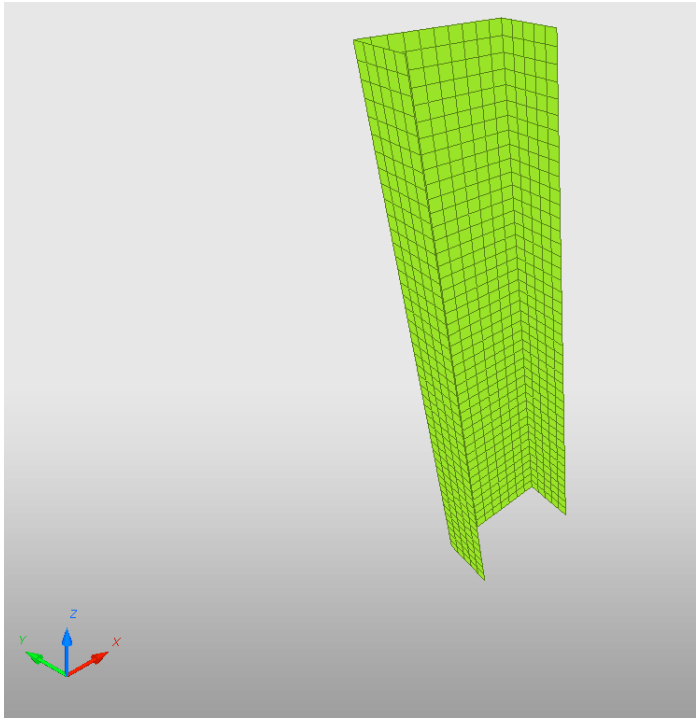




유한요소해석 모델 불러오기



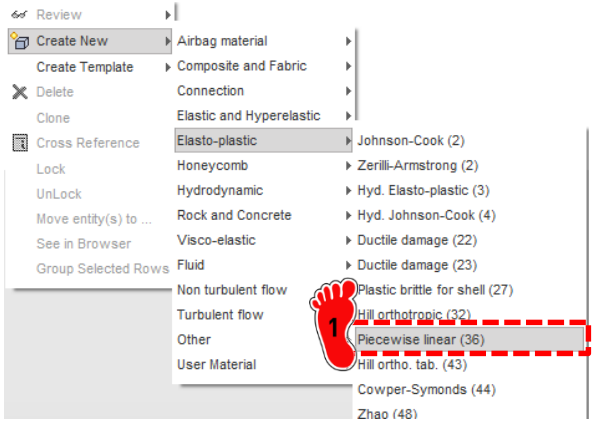




1 File > Import > Nastran 클릭

2 모델 파일 클릭 후 불러오기 (Model_Beam_Crush.fem)

재료 물성 입력 (1)



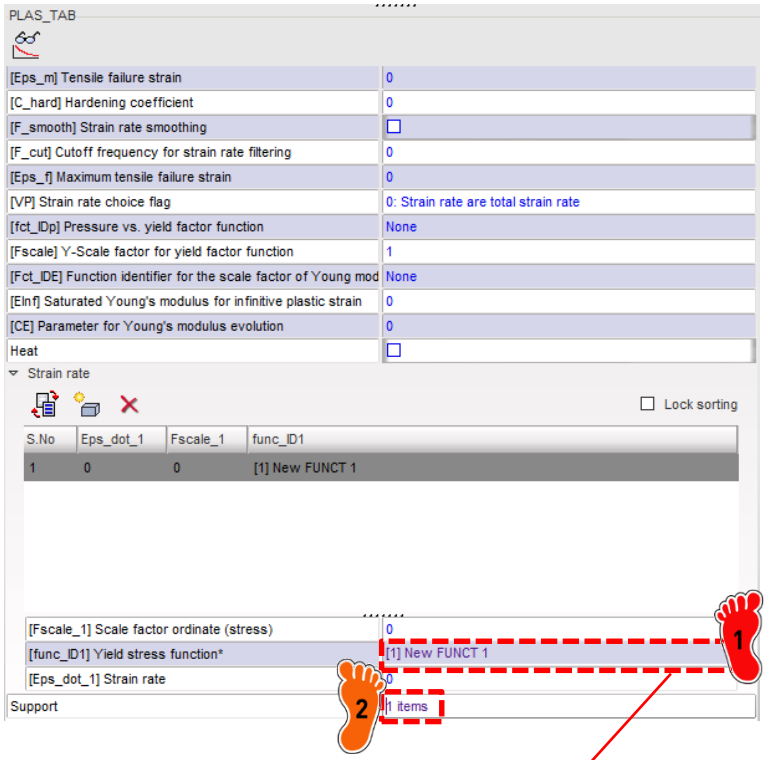
PLAS_TAB

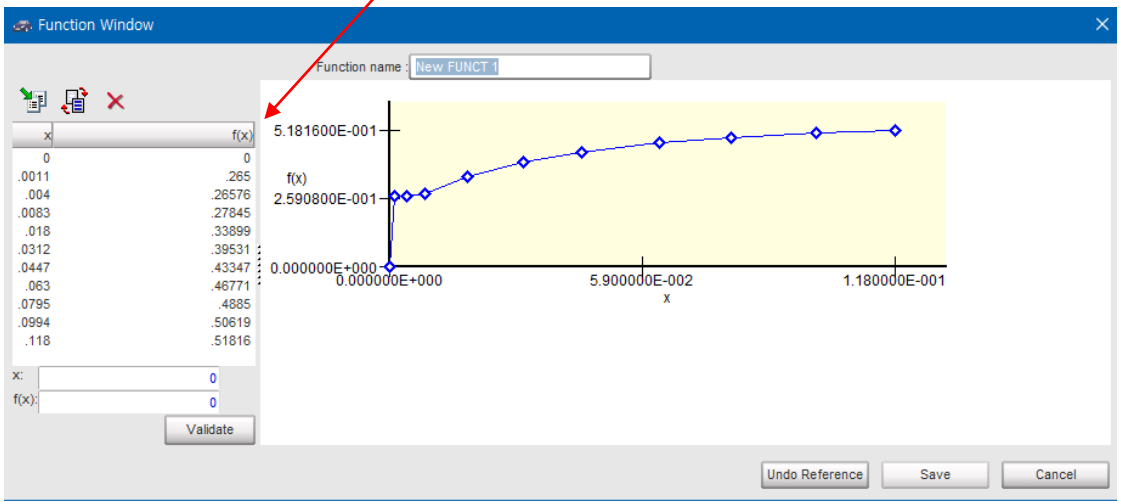
ID	2
Title	New MAT 2
Local Unit System	None
[RHO_I] Initial density*	7.85E-6
[RHO_0] Reference density	0
[E] Young's modulus*	240
[Nu] Poisson's ratio*	.3
[Eps_p_max] Failure plastic strain	0
[Eps_t] Tensile failure strain	0
[Eps_m] Tensile failure strain	0
[C_hard] Hardening coefficient	0
[F_smooth] Strain rate smoothing	☐
[F_cut] Cutoff frequency for strain rate filtering	0
[Eps_f] Maximum tensile failure strain	0
[VP] Strain rate choice flag	0: Strain rate are total strain rate
[fct_IDp] Pressure vs. yield factor function	None
[Fscale] Y-Scale factor for yield factor function	1
[Fct_IDE] Function identifier for the scale factor of Young mod	None
[EInf] Saturated Young's modulus for infinitive plastic strain	0
[CE] Parameter for Young's modulus evolution	0
Heat	☐

1 Piecewise linear(36) 재료 생성

2 재료 물성치 입력
[RHO_I]: 7.85e-6
[E]: 240
[Nu]: 0.3

재료 물성 입력 (2)

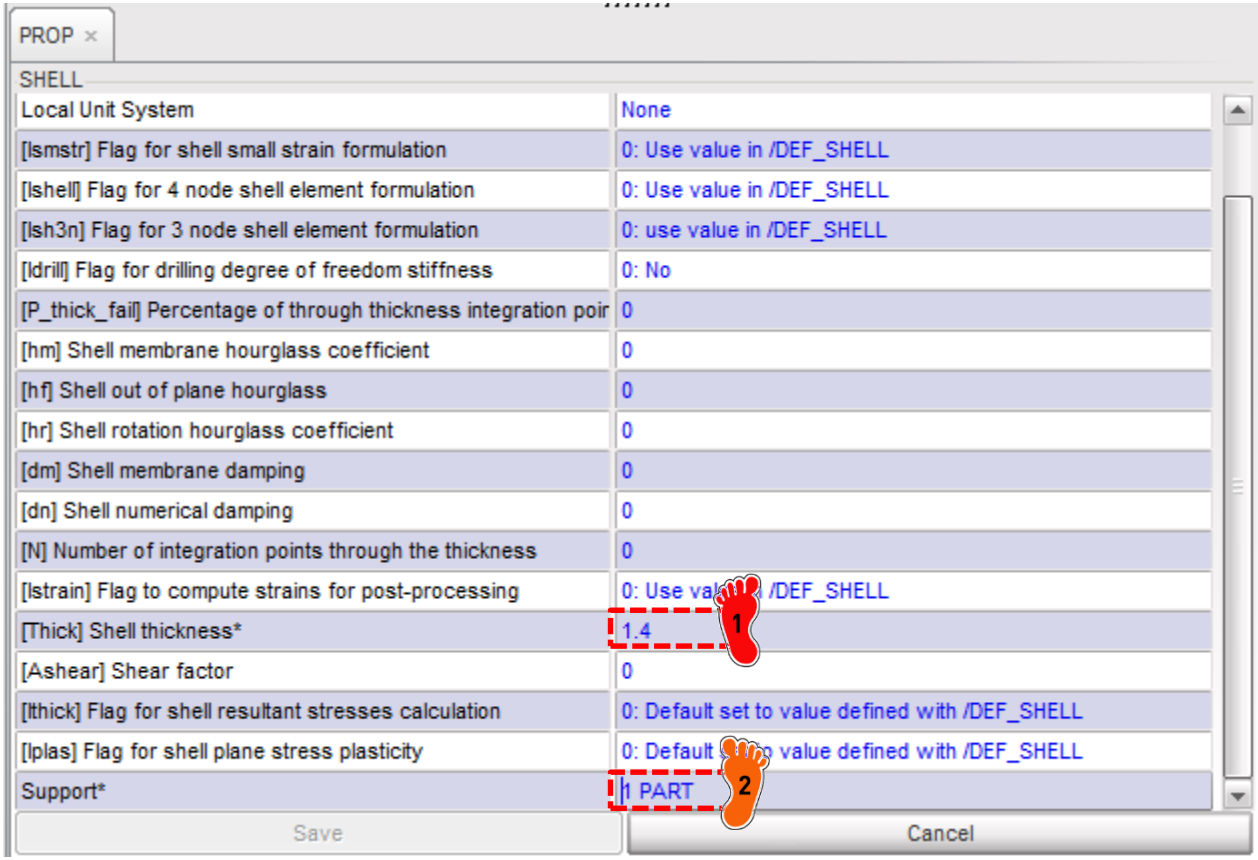




1 Stress-Strain curve 입력

2 모델에 재료 적용

특성 입력



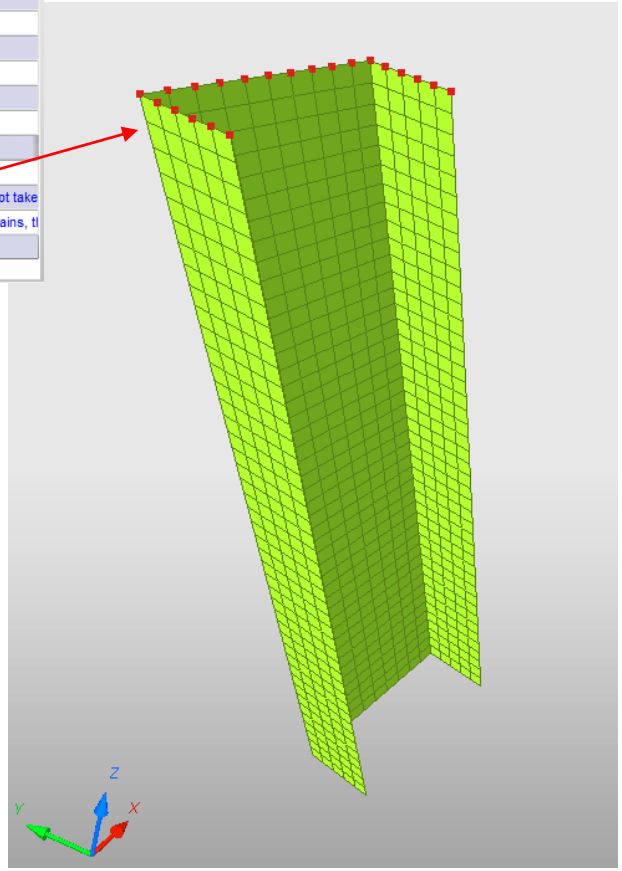
1 쉘 property 생성 후 두께 1.4mm 입력

2 모델에 특성 적용

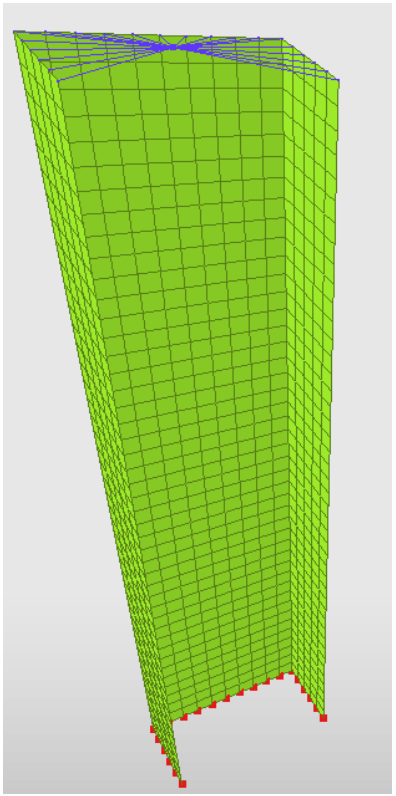
강체 요소 생성

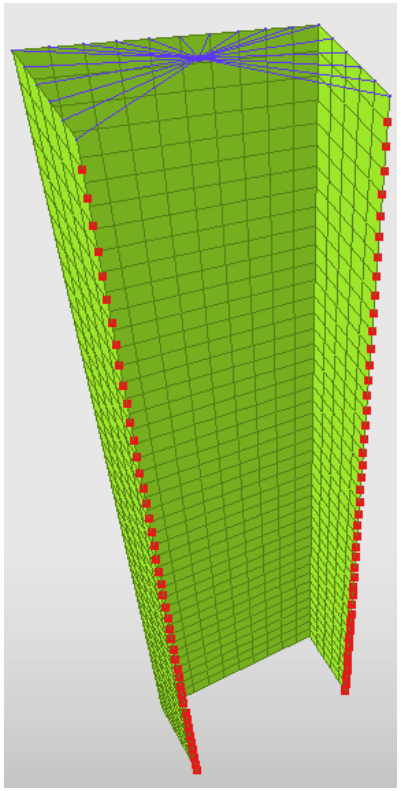
1 Rigid body 생성

RBODY	
Local Unit System	None
Master node in time History	☐
Lagrange multipliers	☐
Automatic master node	☑
Flag for activate/deactivate in run	0: Not ON/OFF in current Engine file (or docked)
[MASS] Mass	0
[Jxx] JXX inertia	0
[Jyy] JYY inertia	0
[Jzz] JZZ inertia	0
[Jxy] JXY inertia	0
[Jyz] JYZ inertia	0
[Jxz] JXZ inertia	0
[ISPHER] Inertia	0: Default, set to 2
[ISENS] Sensor	None
[NSKEW] Skew	None
[IKREM] Keep slave nodes in rigid wall	☐
[ICOG] Center of gravity computation	0: Default, set to 1
[loptoff] Manage domain decomposition of rigid body for RADI	0: CPU cost of elements associated with rigid body is not take
[lexpams] Manage connection through the rigid body between	0: if the rigid body overlaps the AMS and non AMS domains, tl
[Gnod_id] Slave nodes*	21 items
▸ [Gnod_id] Slave nodes (Advanced selector)	



구속조건 설정 (1)





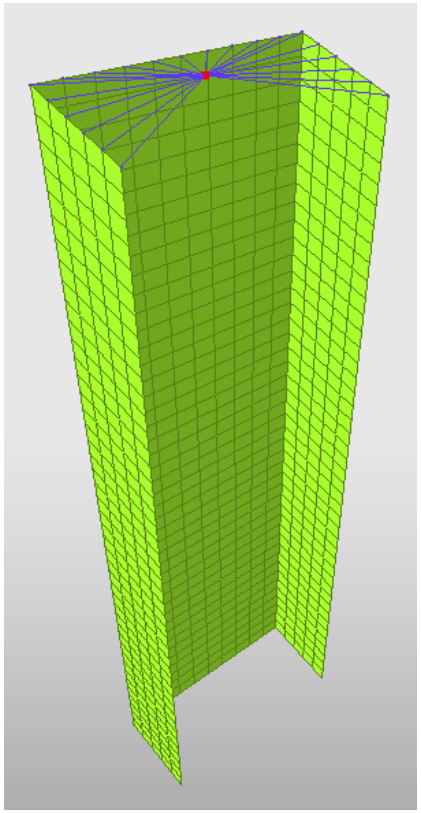
1 빔의 끝부분에 경계조건 설정
(핀구속)

2 빔의 잘린 부분은 대칭 조건으로 경계조건 설정

BCS	
ID	1
Title	New BCS 1
ALE formul.	☐
Lagrange mult. formul.	☐
Translation	
[TX] X translation	☑
[TY] Y translation	☑
[TZ] Z translation	☑
Rotation	
[RX] X rotation	☐
[RY] Y rotation	☐
[RZ] Z rotation	☐
[Skew_id] Skew	None
[Gnod_id] Support*	21 items

BCS	
ID	2
Title	New BCS 2
ALE formul.	☐
Lagrange mult. formul.	☐
Translation	
[TX] X translation	☐
[TY] Y translation	☑
[TZ] Z translation	☐
Rotation	
[RX] X rotation	☑
[RY] Y rotation	☐
[RZ] Z rotation	☑
[Skew_id] Skew	None
[Gnod_id] Support*	88 items
[Gnod_id] Support (Advanced selector)	

구속조건 설정 (1)



BCS	
ID	3
Title	New BCS 3
ALE formul.	☐
Lagrange mult. formul.	☐
Translation	
[TX] X translation	☑
[TY] Y translation	☑
[TZ] Z translation	☐
Rotation	
[RX] X rotation	☑
[RY] Y rotation	☑
[RZ] Z rotation	☑
[Skew_id] Skew	None
[Gnod_id] Support*	1 items

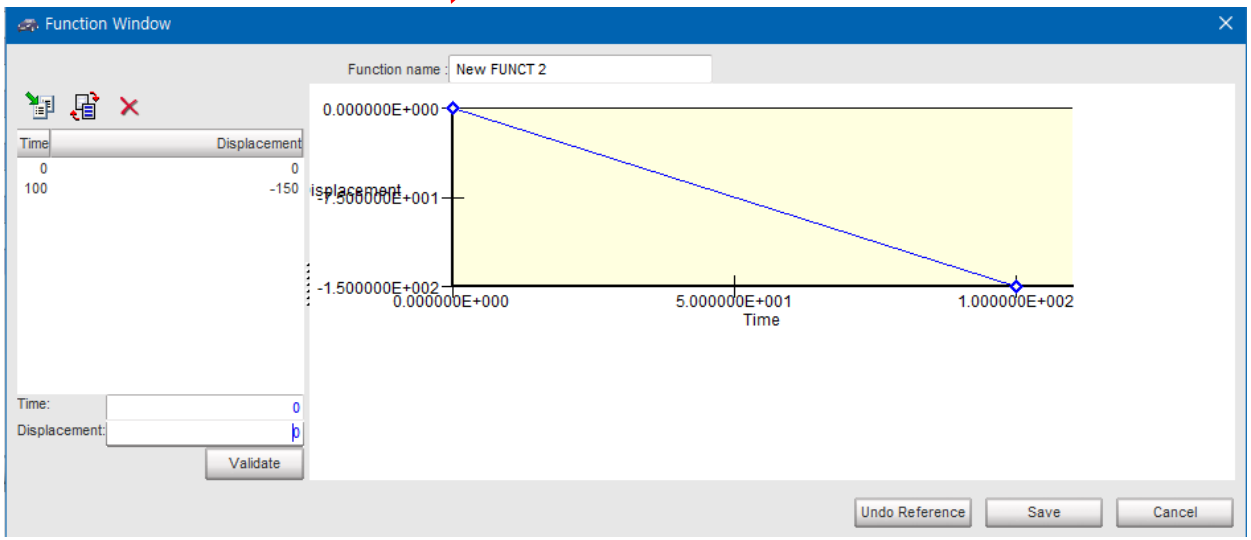
1 강체의 중앙 부분의 절점은 부딪히는 방향의 자유도를 제외한 모든 자유도 구속

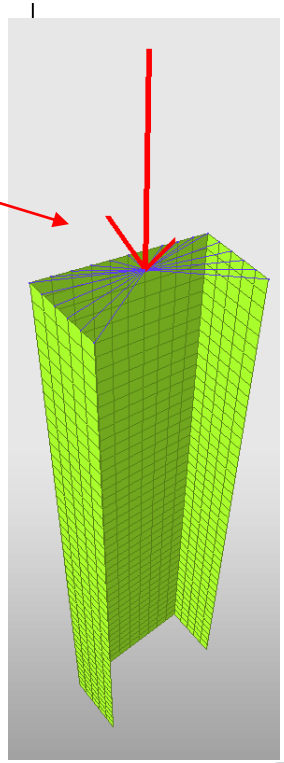
TUBE 변위 설정

IMPDISP	
ID	1
Title	New IMPDISP 1
Local Unit System	None
[Tstart] Start time	0
[Tstop] Stop time	0
[DIR] Direction*	Z: (translation)
[Ifunct] Time function*	[2] New FUNCT 2
[Scale_x] X-Scale factor	1
[Scale_y] Y-Scale factor	1
[Iskew] Skew	None
[Isensor] Sensor	None
[Frame] Frame	None
[Icoor] Type of usage of the coordinate system	Cartesian coordinates
[Gnod_id] Support*	[1] IMPDISP_group_7_of_NODE

1 [DIR]: Z (translation) 선택
[Ifunct]: 생성

2 Rigid body의 마스터 절점에 적용





접촉조건 설정

TYPE7	
ID	1
Title	New INTER 1
Self Impact	☑
Local Unit System	None
Formulation	0: Classical
[Istf] Stiffness definition	0: Default, set to value defined in /DEFAULT/INTER/TYPE7
[Ithe] Heat contact	☐
[Igap] Gap/element option	0: Default, set to value defined in /DEFAULT/INTER/TYPE7
[Fpenmax] Maximum fraction of initial penetration	0
[Ibag] Vent hole closure when contact	0: Default, set to value defined in /DEFAULT/INTER/TYPE7
[Idel] Node and segment deletion	0: Default, set to value defined in /DEFAULT/INTER/TYPE7
[Icurv] Slave gap with curvature	0: No curvature
[Iadm] Local curvature flag	0: Not activated
[Stfac] Scale factor for stiffness	1
[dtmin] Limiting nodal time step	0
[Irem_gap] Flag for deactivating slave nodes if element size < ga	0: Default, set to value defined in /DEFAULT/INTER/TYPE7
[Irem_i2] Flag for deactivating the slave node, if the same conta	0: default, set to the value defined in /DEFAULT/INTER/TYPE7
[Gapmin] Min. gap for impact activ.	.9
▷ [I_BC] Bound. cond. deactivation	
[Inacti] Stiffness deactiv. (init. penetration)	0: Default, set to value defined in /DEFAULT/INTER/TYPE7
[VisS] Critical damping coeff. on interface stiffness	0
[VisF] Critical damping coeff. on interface friction	0
[Tstart] Start time	0
[Tstop] Stop time	0
[Bumult] Sorting factor	0
[Ifric] Friction formulation	0: Static Coulomb
[Fric] Coulomb friction	.2
[Iform] Friction penalty formulation	0: Default, set to value defined in /DEFAULT/INTER/TYPE7
[Ifiltr] Friction filtering	0: No filtering
[sens_ID] Sensor to Activate/Deactivate the interface	None
[fct_IDf] Friction coefficient with temperature function identifier	None
[AscaleF] Abscissa scale factor on FCT_IDK	0
[fric_ID] Friction identifier for friction definition for selected pairs	None
[Mast_id] Master surface*	1 items

1. Contact interface 생성 후 파라미터 설정
   Self Impact: 체크
   [Stfac]: 1
   [Gapmin]: 0.9
   [Fric]: 0.2

2. 접촉 파트 설정

해석 케이스 설정

1 Control card 설정

RUN_NAME_NUMBER_LETTER

RUN_NAME_NUMBER_LETTER	
Run Number	1
[DT_STOP] Final time for run	4.01
Restart letter	: Default

ANIM_DT

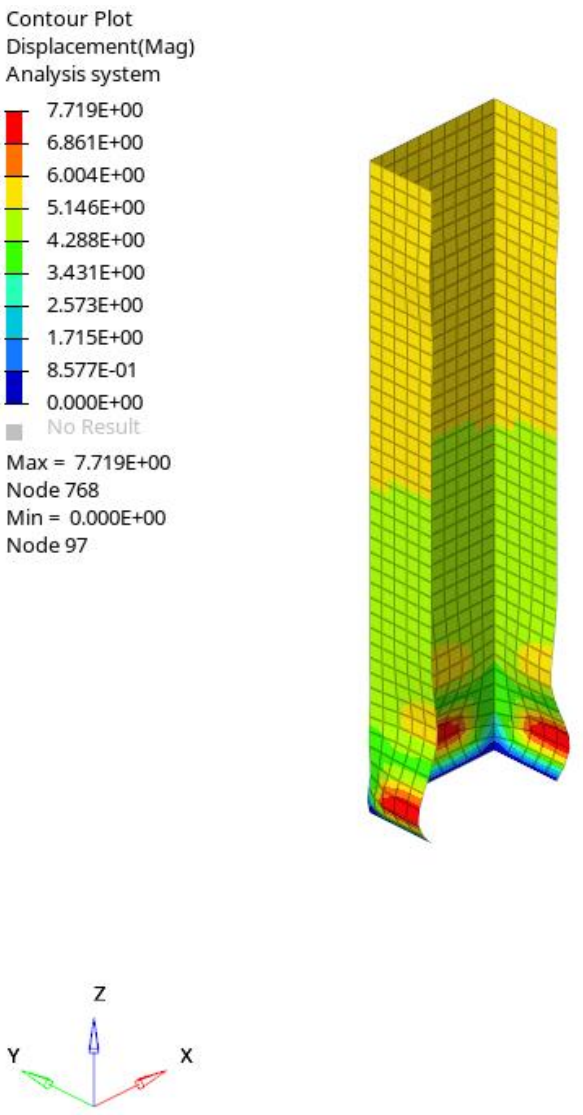
ANIM_DT	
[TSTART] Start time	0
[TFREQ] Time frequency	0.01

ANIM_VECT

S.No	Variable name to be saved in animation file
1	FOPT

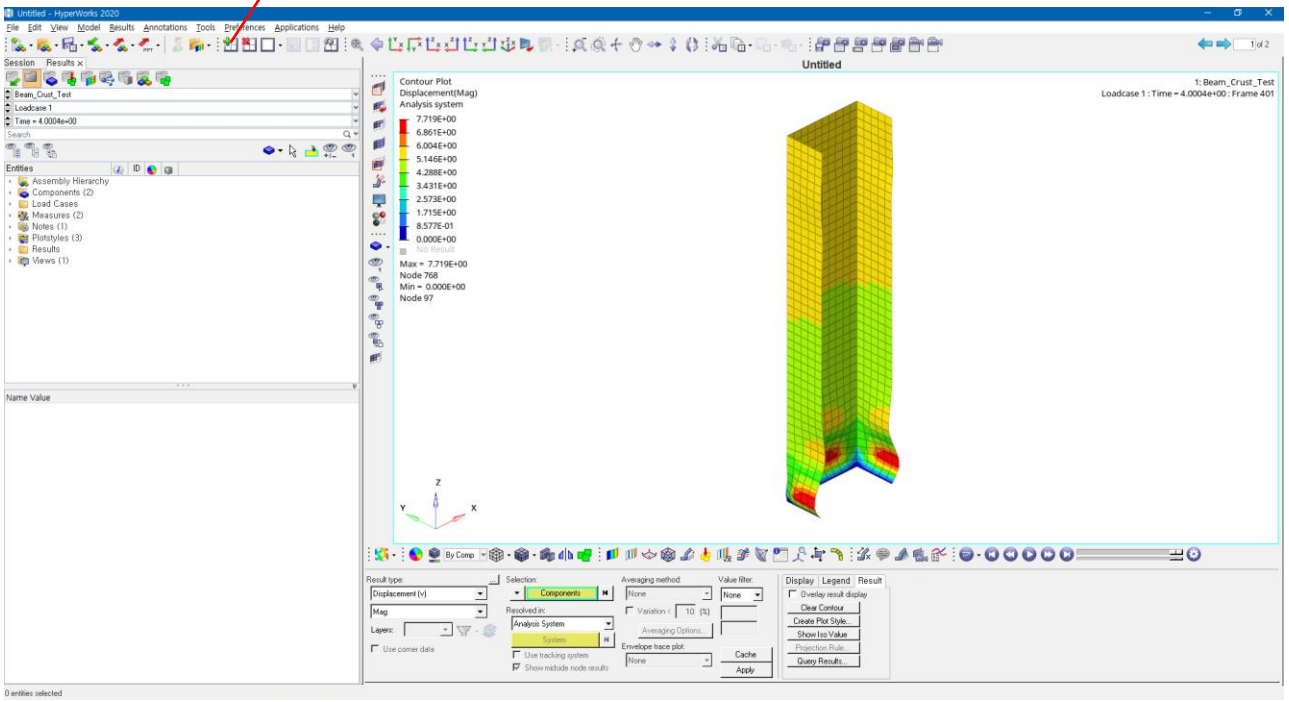
ANIM_VECT_RESTYPE_SUPPORT	
Variable name to be saved in animation file*	FOPT: Forces and moments for rigid bodies, rigid walls and sect

후처리 (1)



1 모델 export 후 RADIOSS 에서 해석 실행

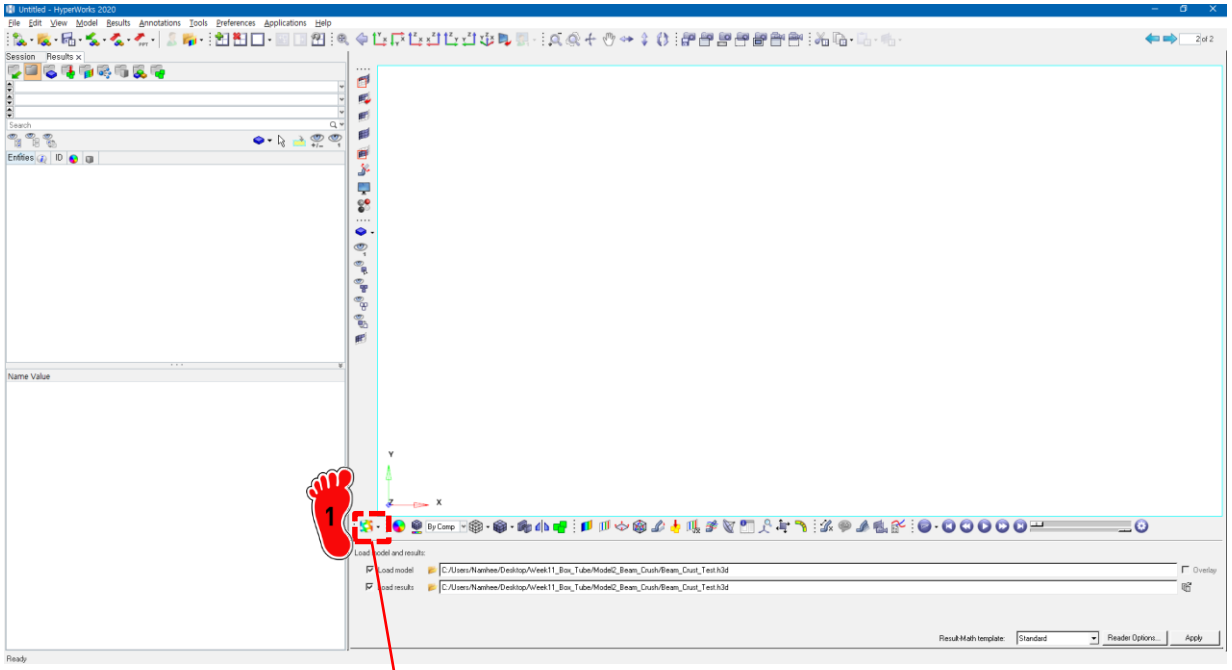
후처리 (2)

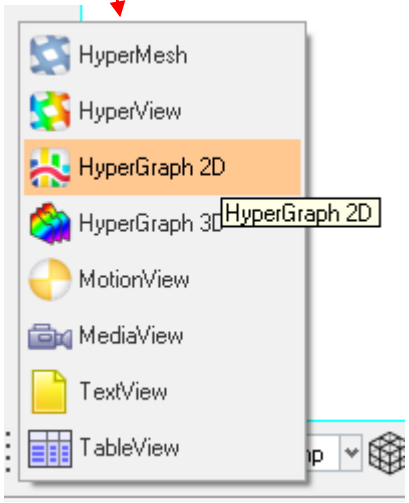


1 HyperView에서 Add Page 버튼 클릭

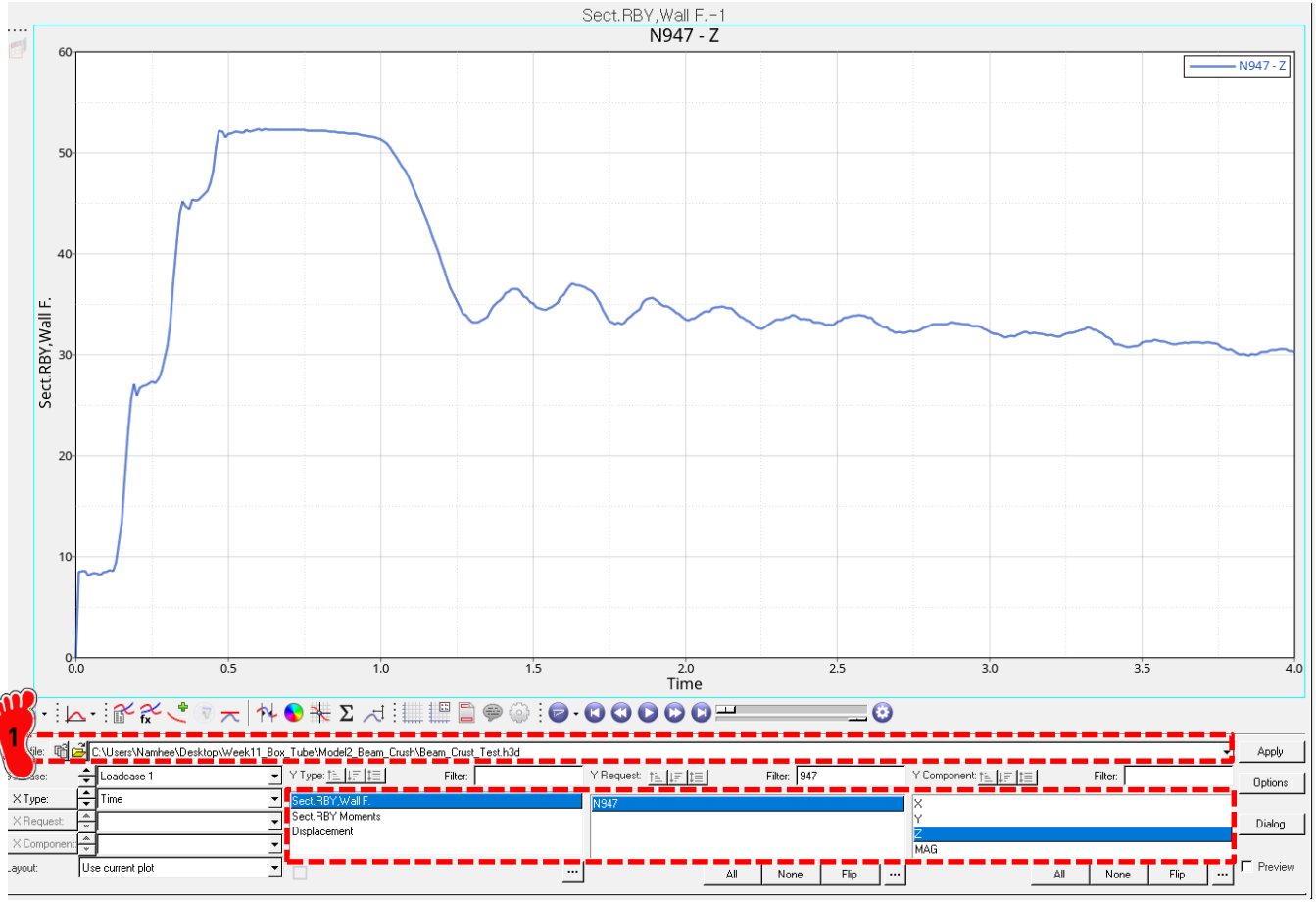
후처리 (2)

1 HyperGraph 2D 클릭





후처리 (2)



1 해석 결과 파일 불러오기 (*.h3d 파일)

2 Sect.RBY, Wall F. 선택, 947번 절점 선택, Z 방향으로의 그래프 확인 (Rigid element의 마스터 절점의 충돌 하중 확인)

후처리 (3)

Analytic solution

$$P_M = 386t^{1.86}b^{0.14}\sigma_Y^{0.57} = 30237 \text{ N}$$

$$P_{MAX} = 2.87P_M = 86780 \text{ N}$$

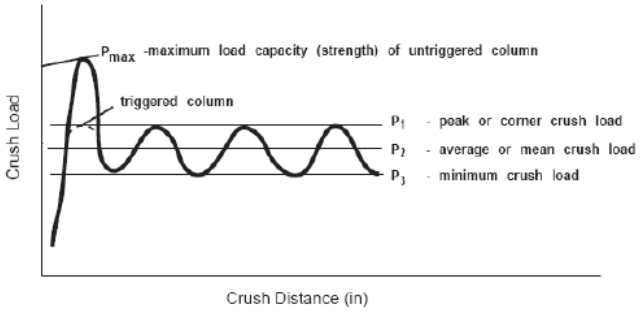
$$P_1 = 1.42P_M = 42936 \text{ N}$$

$$P_2 = 0.57P_M = 17235 \text{ N}$$

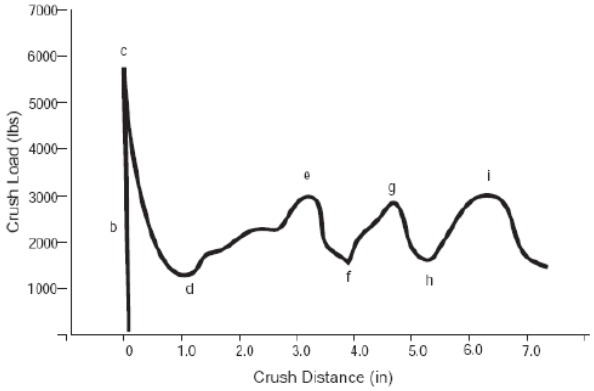
FEM solution

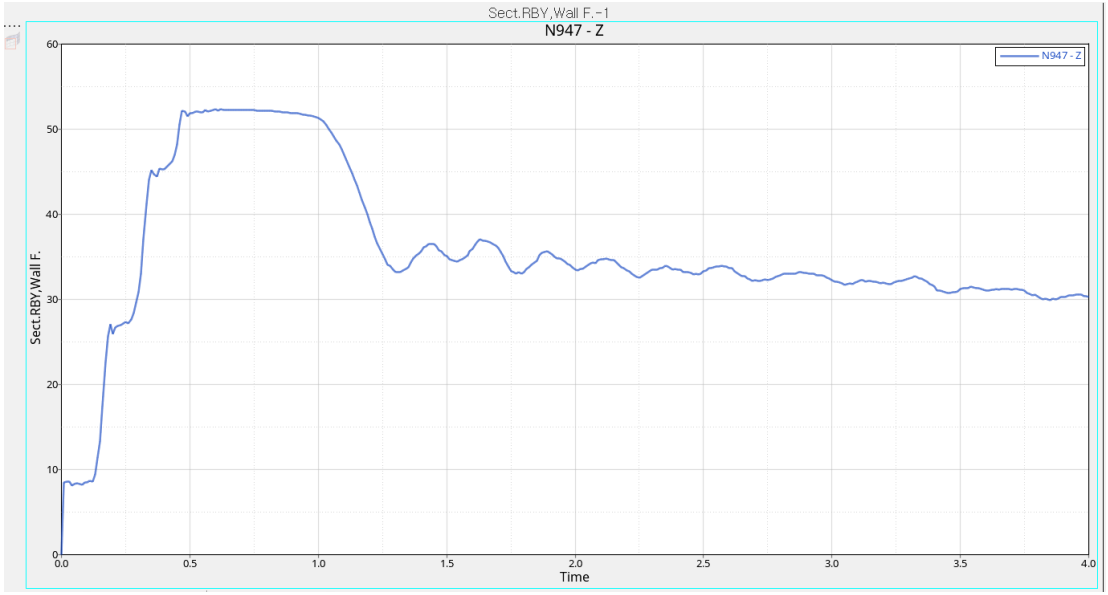
$$P_{MAX} = 52128 \text{ N}$$

Ideal graph



Practical graph





숙제

예제에서 수행한 정 사각형의 정적 충돌 하중을 100%로 놓고, 나머지 형상 중 하나를 결정하여 평균 정적 충돌 하중을 구하시오.

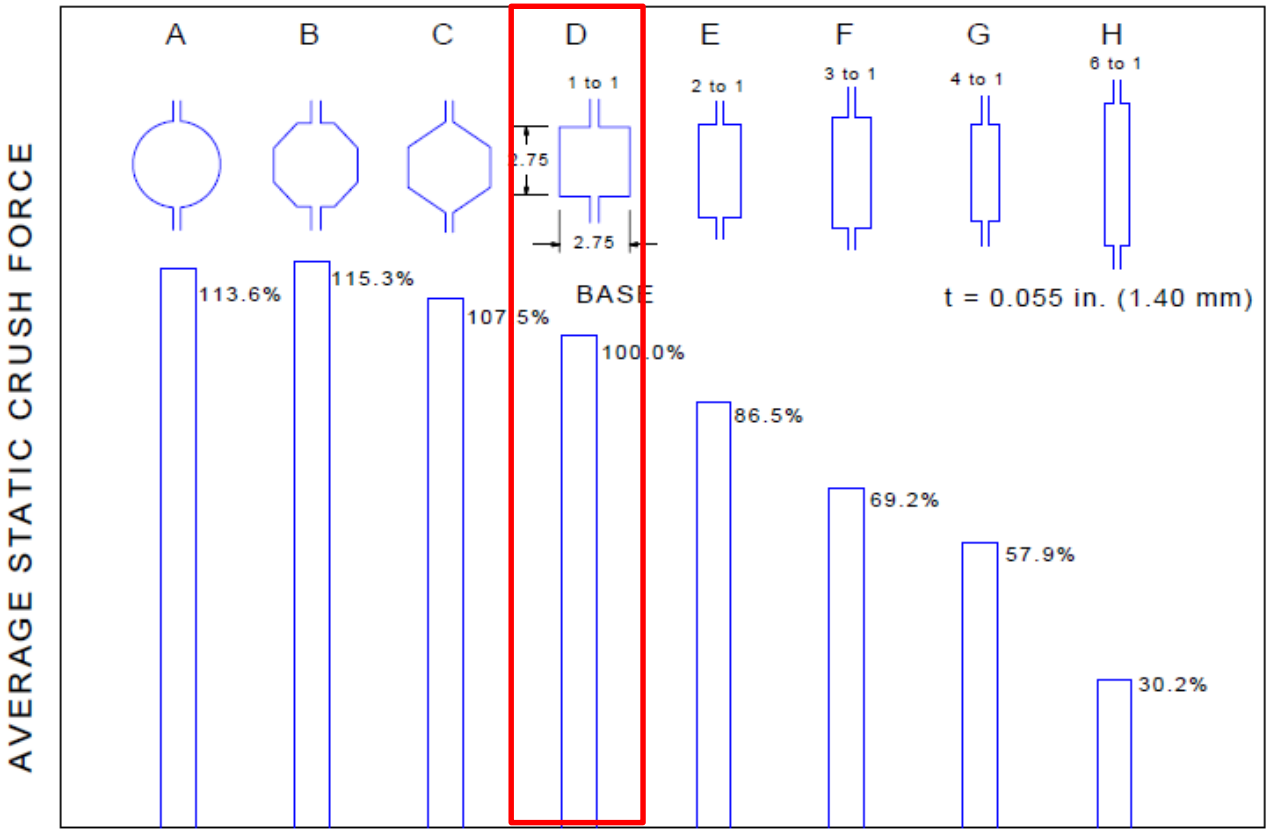


Figure 3.6.4-1 Average static crush force vs. section shape
(All samples were the same mass and length)