Explicit Dynamic Analysis (1)

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

OUTLINE

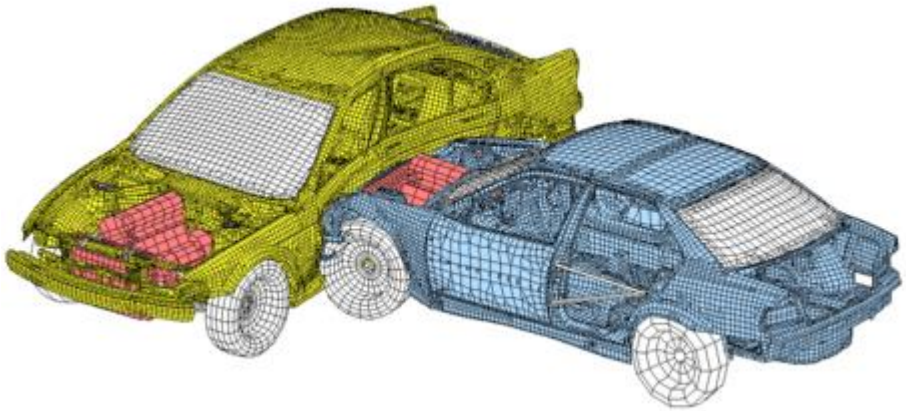
- **Lecture Goals**
  - ✓ Tube(shell 요소)모델의 외연적 동적 해석(explicit dynamic analysis)을 수행하고 충돌 결과를 분석한다.

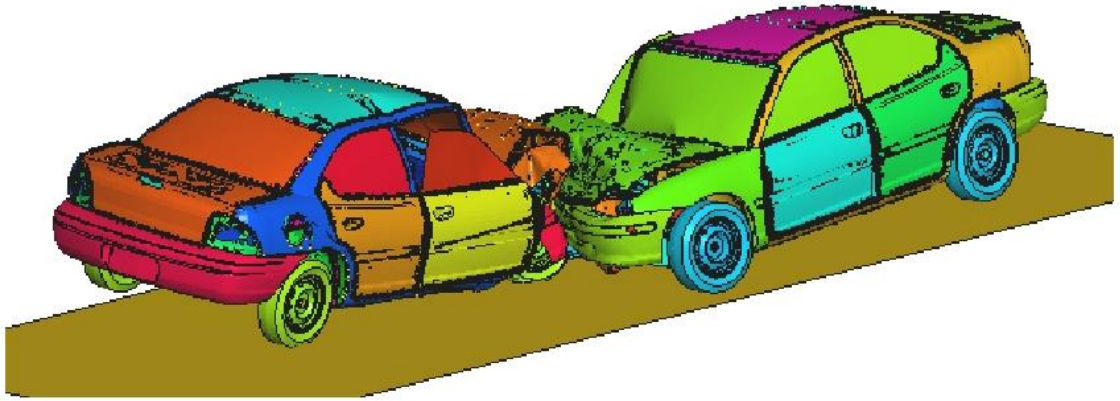
- **Contents**
  - ✓ Tube impact analysis (RADIOSS example)
  - ✓ Axial crush load of square section

- 해석 프로세스
  - 기하형상 생성
  - 재료 물성 및 특성 입력
  - 요소망 생성
  - 구속조건 설정
  - 하중조건 설정
  - 해석케이스 정의 및 해석 실행
  - 후처리

INTRODUCTION

Crash analysis of body structure





INTRODUCTION

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

내연적(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

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

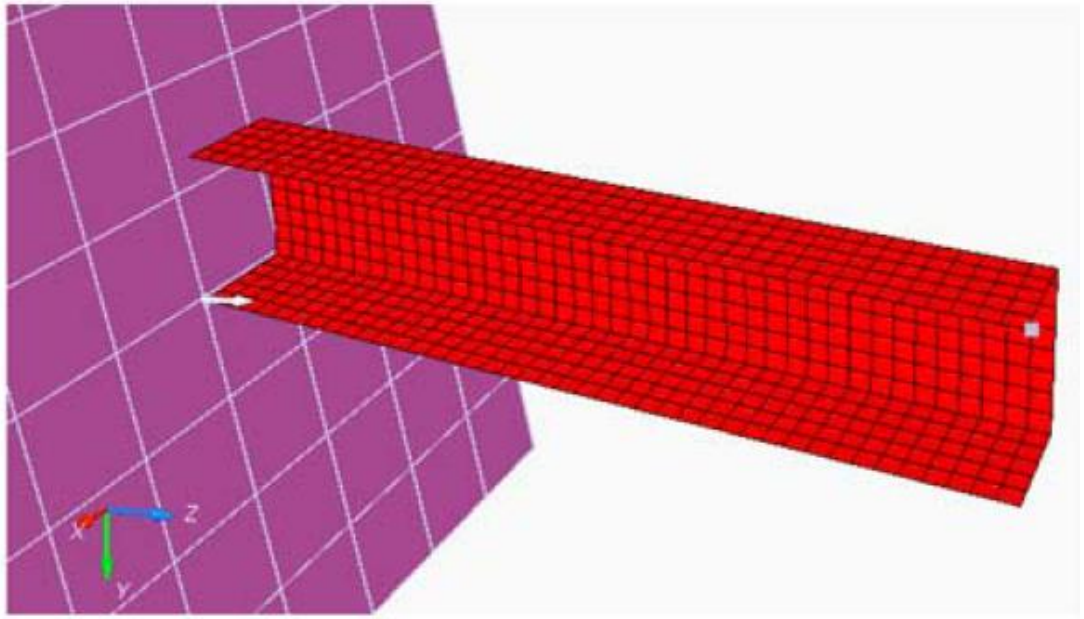
$$x_{n+1} = f(x_n, \dots)$$

TUBE IMPACT ANALYSIS (RADIOSS EXAMPLE)

예제: TUBE IMPACT EXAMPLE

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.

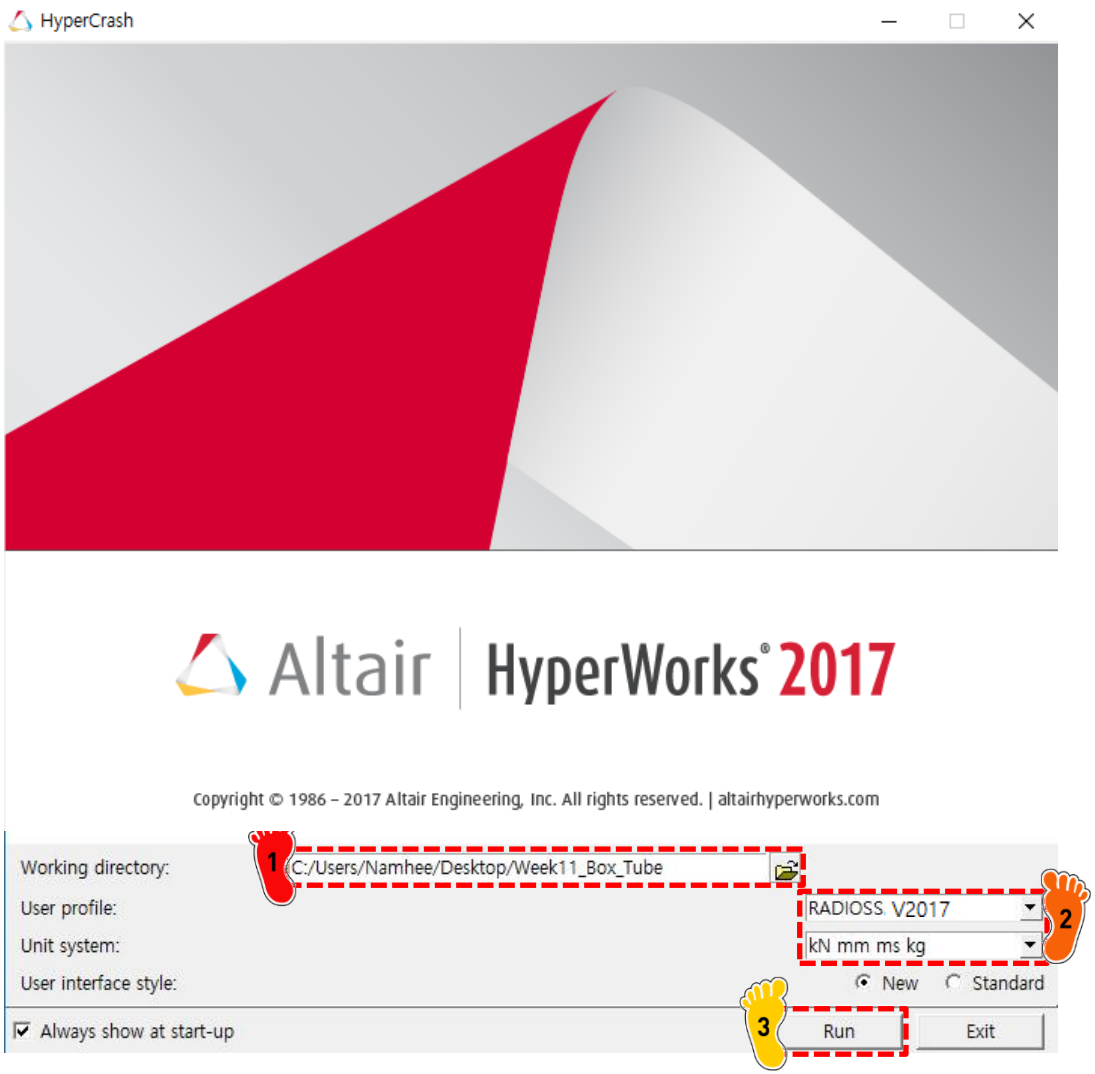


Model

- The tube thickness is 0.914 mm.

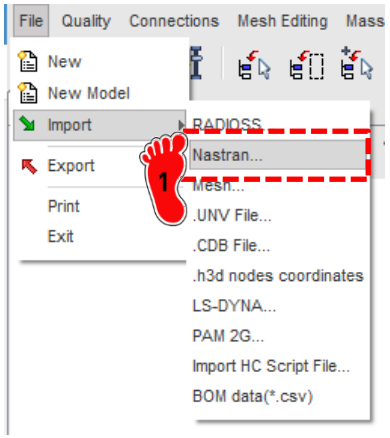
- $\rho = 7.85e^{-6}$ Kg/mm^3 — Initial density
- E = 210 GPa — Young's modulus
- v = 0.33 — Poisson coefficient
- σ_0 = 0.206 GPa — [a] Yield Stress

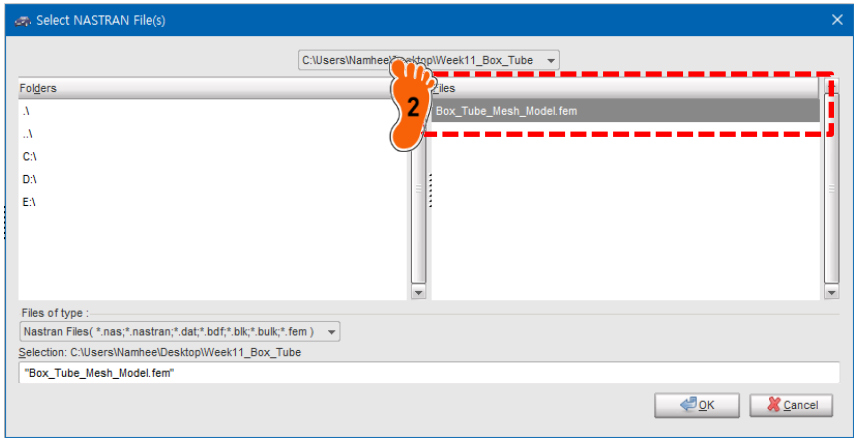
HYPERCRASH 실행

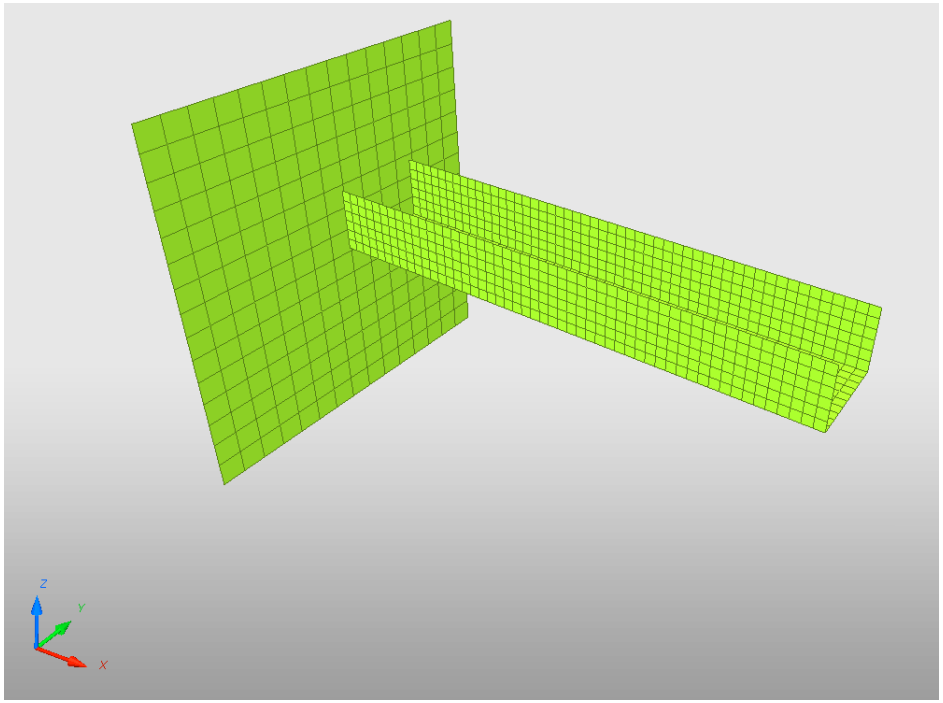


1. HyperCrash2017 실행 후 Working directory 설정 (모델 파일이 있는 폴더로 지정)
2. User profile, Unit system 설정
3. Run 클릭

유한요소해석 모델 불러오기







1 File > Import > Nastran 클릭

2 모델 파일 클릭 후 불러오기 (Box_Tube_Mesh_Model.fem)

재료 물성 입력

HyperCrash Elasto-plastic 재료의 종류 (/MAT/LAW#)

- Johnson-Cook (2)
- Zerilli-Armstrong (2)
- Hyd. Elasto-plastic (3)
- Hyd. Johnson-Cook (4)
- Ductile damage (22)
- Ductile damage (23)
- Plastic brittle for shell (27)
- Hill orthotropic (32)
- Piecewise linear (36)
- Hill ortho. tab. (43)
- Cowper-Symonds (44)
- Zhao (48)
- Steinberg-Guinan (49)
- Gurson (52)
- Tsai-Wu tabulated (53)
- Barlat (57)
- Piecewise non-linear (60)
- Hansel trip steel (63)
- U&A trip steel (64)
- Elastomer (65)
- Plasticity compression tension (66)
- Anisotropic Hill (72)
- Thermal Hill orthotropic (73)
- Thermal Hill orthotropic for solids (74)
- Semi Analytic Plastic (76)
- Yoshida Uemori (78)
- Johnson-Holmquist (79)
- Hot stamping (80)
- Swift-Voce (84)
- Barlat2000 (87)

- This model expresses material stress as a function of strain, strain rate and temperature.
- Particular type of Mises plasticity model with analytical forms of the hardening law and rate dependence.
- JC is suitable for high-strain-rate deformation of many materials

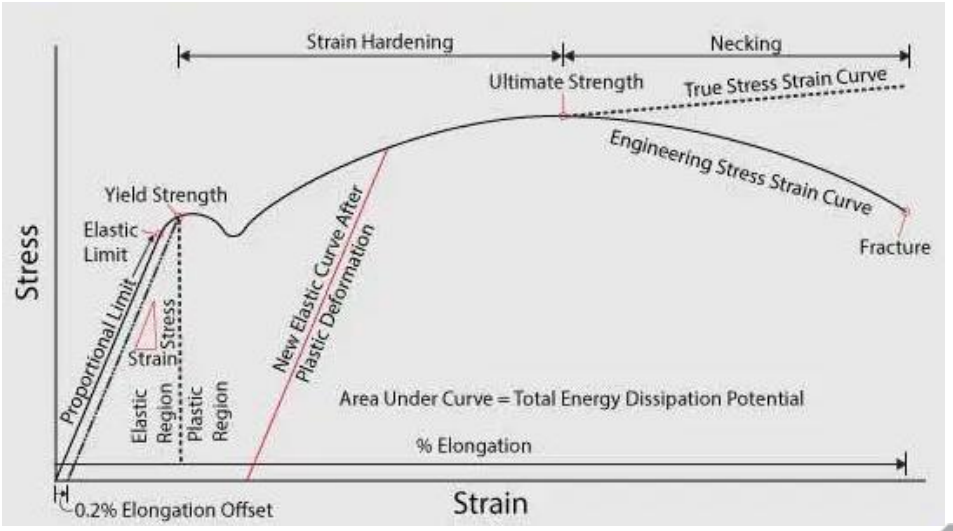
$$\sigma = (a + b\varepsilon_p^n)(1 + c\ln\frac{\dot{\varepsilon}}{\dot{\varepsilon}_0})(1 - T^{*m})$$

Influence of temperature change

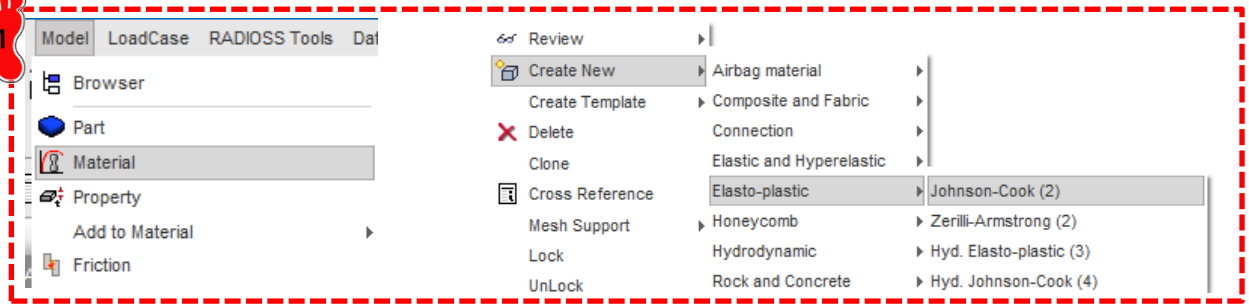
Influence of strain rate

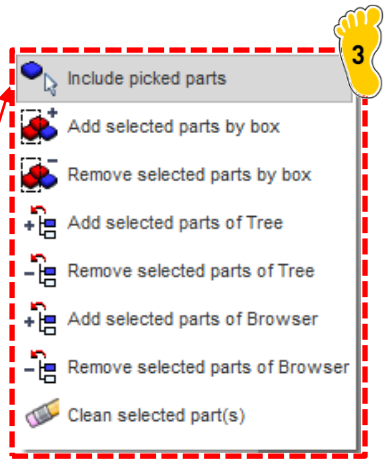
Influence of plastic strain

σ = Stress	n = Hardening Exponent
ε_p = Plastic strain	c = Strain rate coefficient
a = Yield stress	$\dot{\varepsilon}$ = Strain rate
b = Hardening modulus	$\dot{\varepsilon}_0$ = Reference strain rate



재료 물성 입력



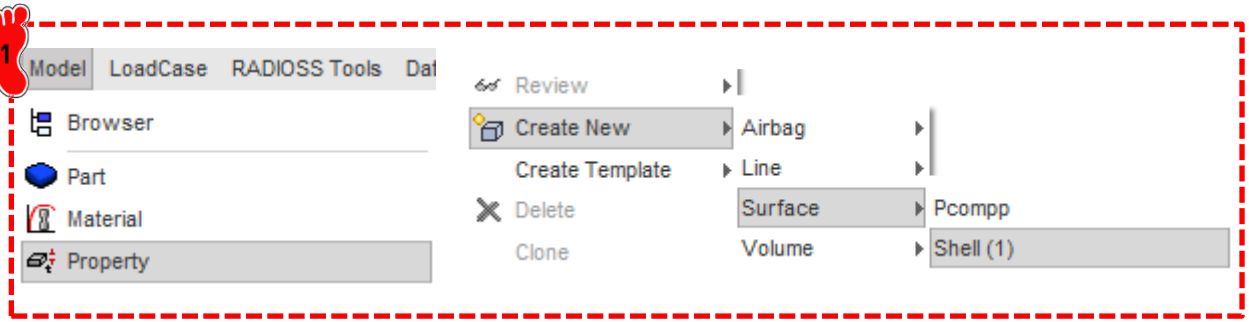


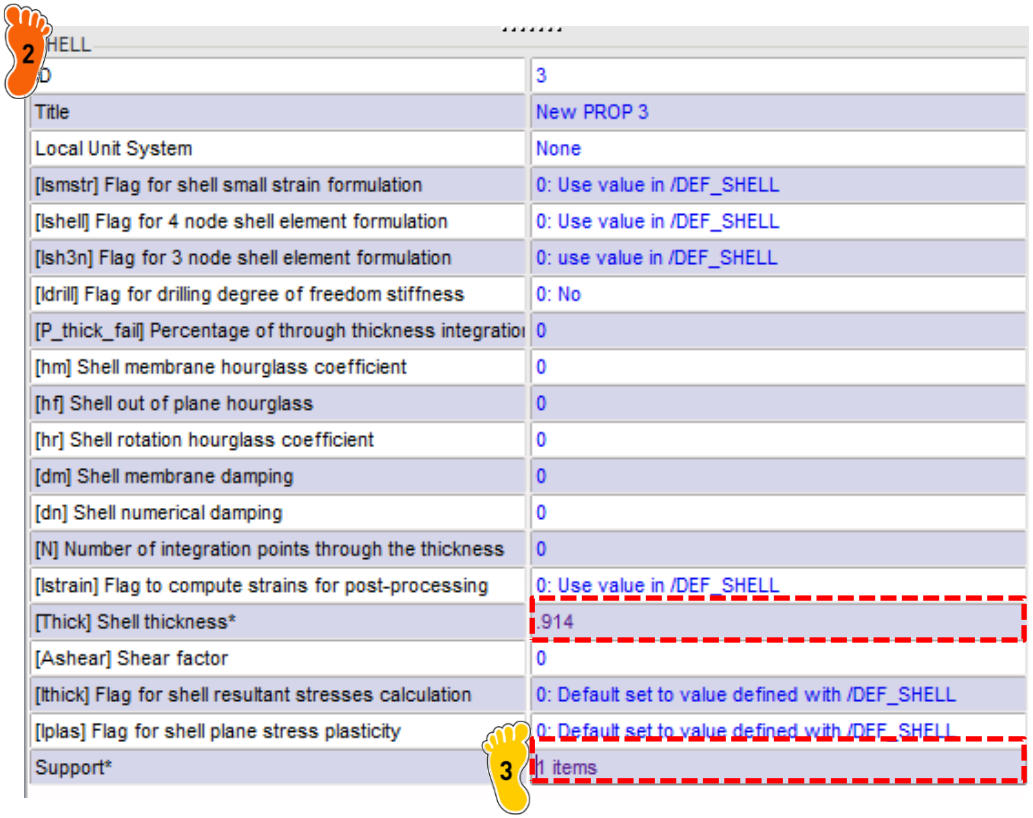
1 Model > Material 클릭, 마우스 우클릭 후 Create New > Elasto-plastic > Johnson-Cook (2) 클릭

2 재료 물성치 입력
[RHO_I]: 7.85e-6 kg/mm^3
[E]: 210 GPa
[Nu]: 0.3
[a]: 0.206 GPa

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

특성 입력





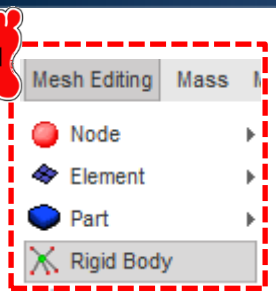
1 Model > Property 클릭, 마우스 우클릭 후 Create New > Surface > Shell (1) 클릭

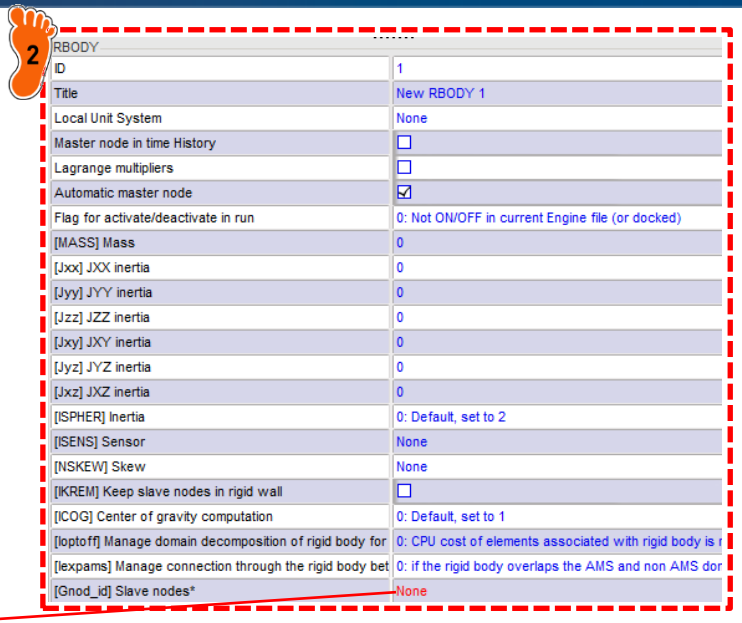
2 재료 물성치 입력 [Thick]: 0.914

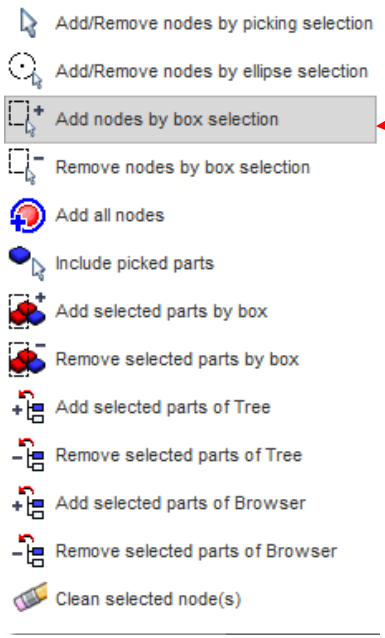
3 Support* 항목의 경우 마우스 우클릭하여 "Select in graphics" 선택, Include picked part ( ) 클릭 후 모델 선택

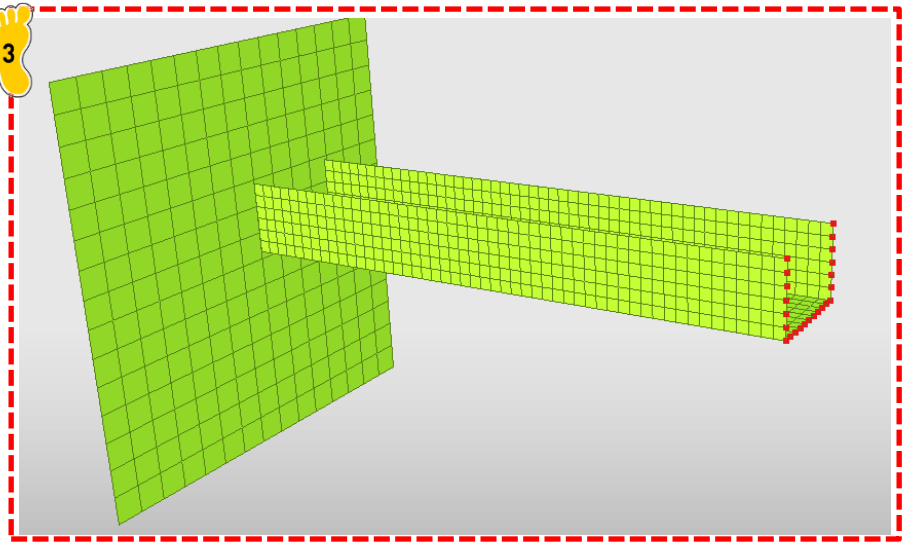
Help: /PROP/TYPE1 (SHELL)

강체 요소 생성







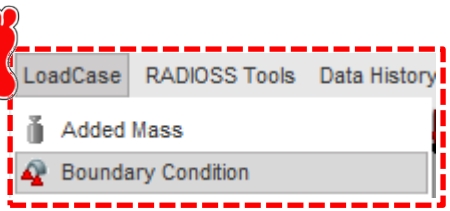


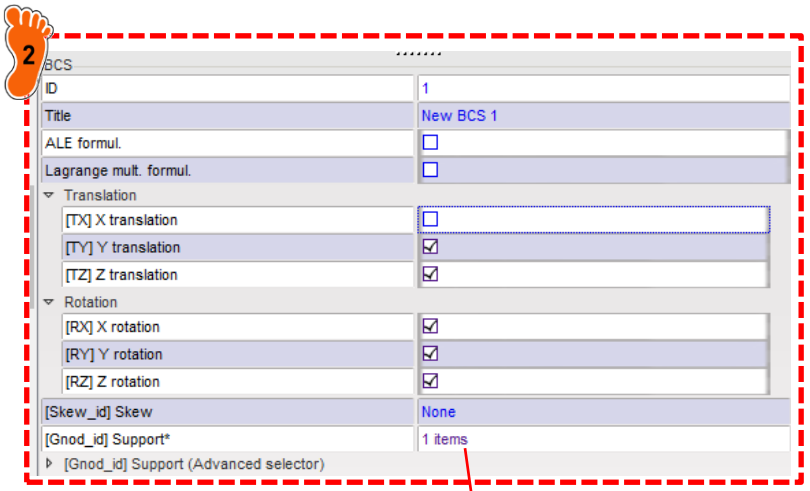
1 Mesh Editing > Rigid Body 클릭, 마우스 우클릭 후 Create New 클릭

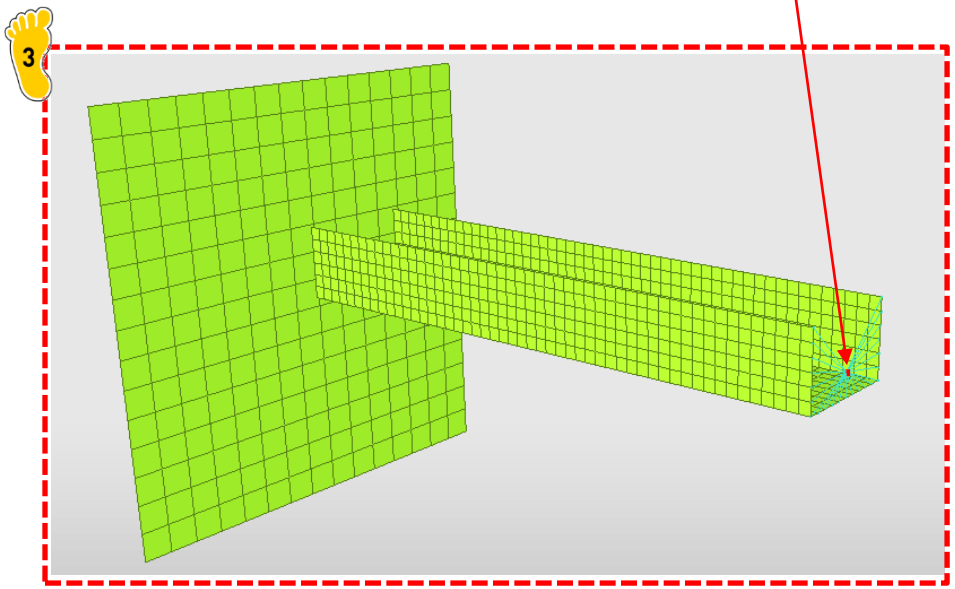
2 Slave nodes 항목에서 마우스 우클릭 후 Select in graphics 선택 > Add nodes by box selection 선택

3 그림과 같이 tube 끝 절점 선택 후 save 클릭

구속조건 설정 (1)





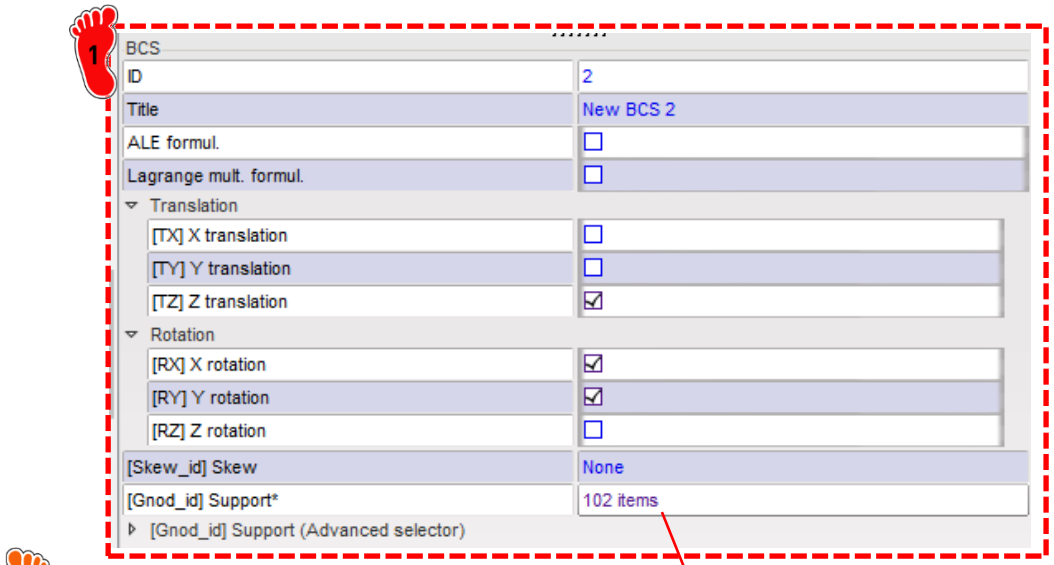


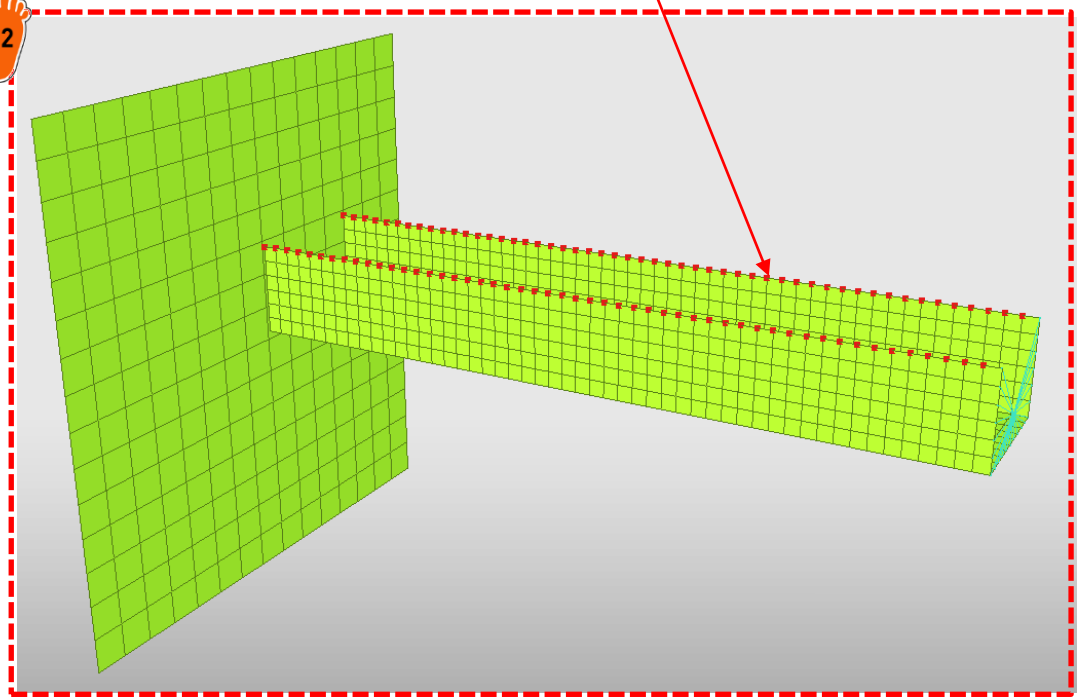
1 LoadCase > Boundary Condition 클릭, 마우스 우클릭 후 Create new 클릭

2 TX 제외하고 모두 체크

3 앞서 제작한 Rigid 요소의 Master node 선택 후 save 클릭

구속조건 설정 (2)





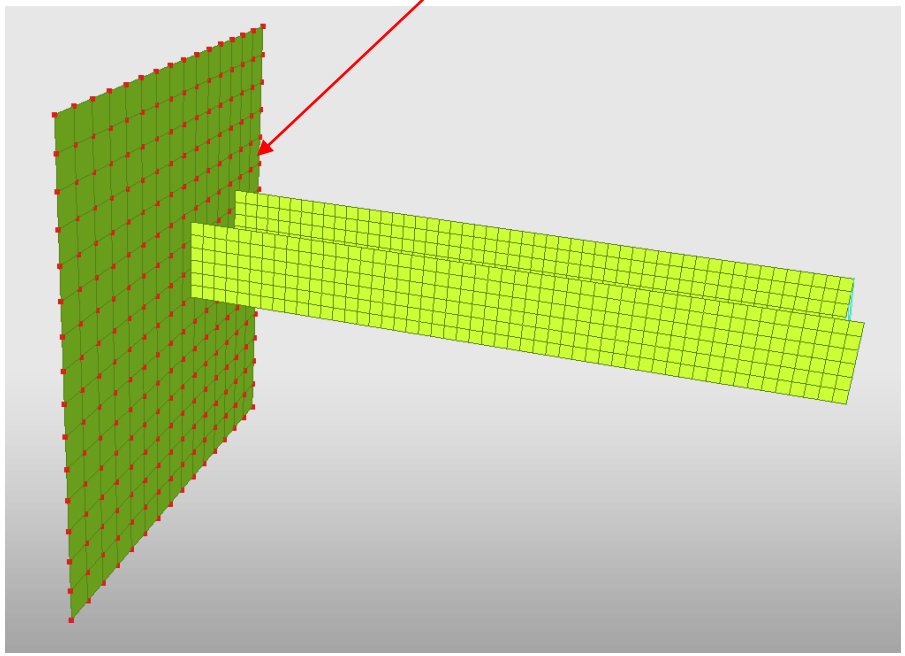
1 경계조건 추가 후 RZ, RX, RY 체크

2 그림과 같이 tube의 상단 절점 선택 후 save 클릭

구속조건 설정 (3)

BCS

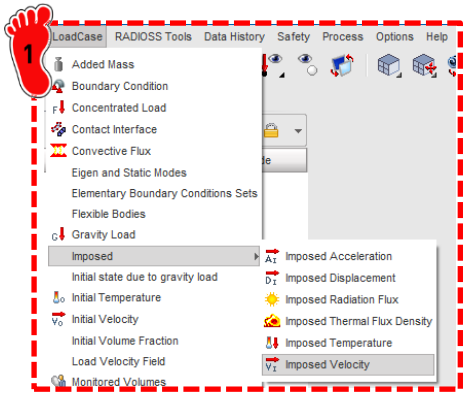
ID	3
Title	New BCS 3
ALE formul.	☐
Lagrange mult. formul.	☐
Translation	
[TX] X translation	☑
[TY] Y translation	☑
[TZ] Z translation	☑
Rotation	
[RX] X rotation	☑
[RY] Y rotation	☑
[RZ] Z rotation	☑
[Skew_id] Skew	None
[Gnod_id] Support*	256 items
[Gnod_id] Support (Advanced selector)	

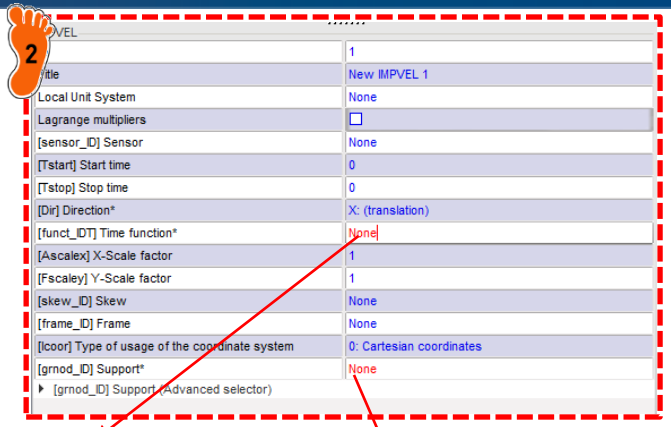


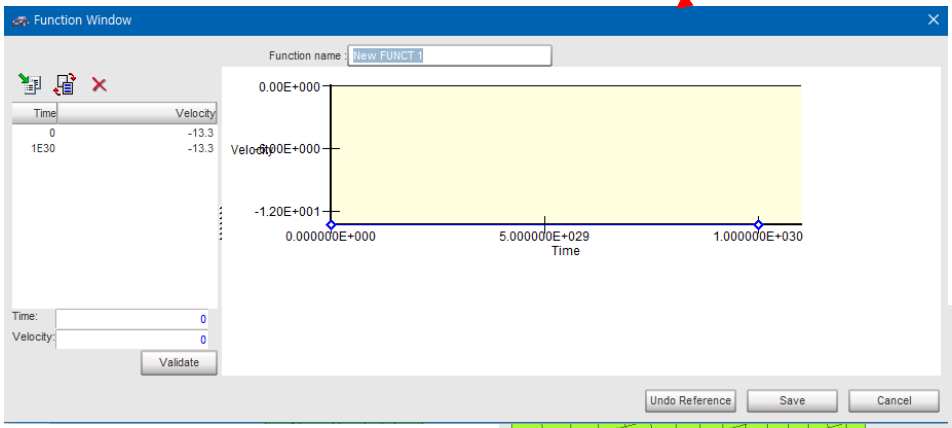
1. 경계조건 추가 후 자유도 모두 체크

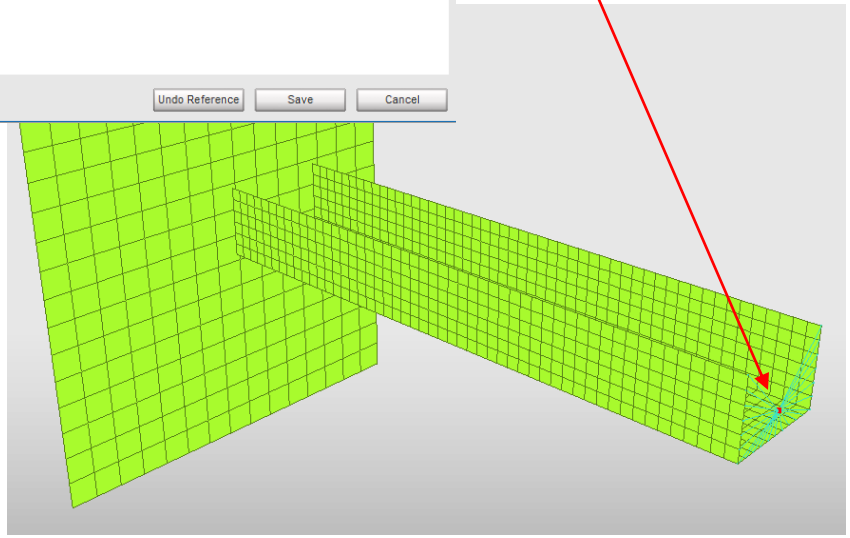
2. 그림과 같이 벽면 절점 선택 후 save 클릭

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

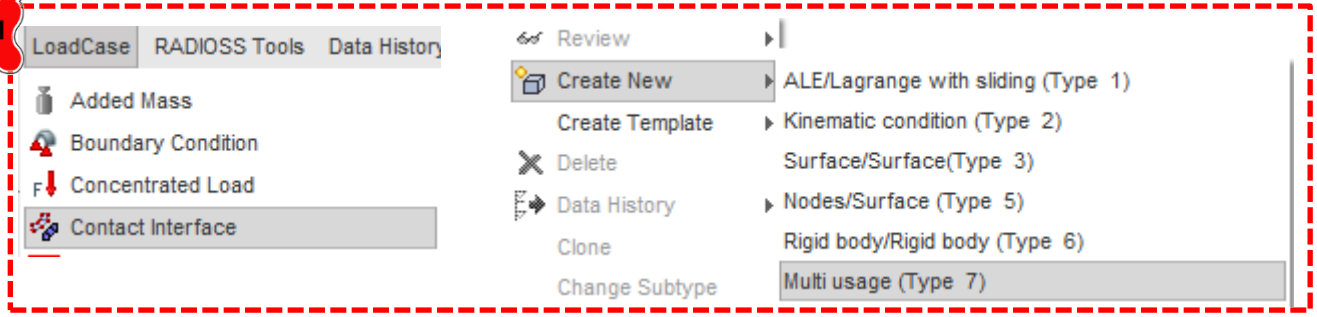
접촉조건 설정

HyperCrash 접촉조건의 종류 (/INTER/TYPE#)

ALE/Lagrange with sliding (Type 1)
Kinematic condition (Type 2)
Surface/Surface(Type 3)
Nodes/Surface (Type 5)
Rigid body/Rigid body (Type 6)
Multi usage (Type 7)
Drawbeads (Type 8)
ALE/Lagrange with Void (Type 9)
Tied with void(Type 10)
Edge to edge (Type 11)
Fluid/Fluid (Type 12)
Ellips. surface/Node (Type 14)
Ellips. surface/Surface (Type 15)
Node/Solid (Type 16)
Solid/Solid (Type 17)
Fluid/Structure (Type 18)
Surface/Surface with edge treatment (Type 19)
Multi usage constant stiffness (Type 20)
Surface/Rigid surface (Type 21)
Airbag surface (Type 23)
Surface/Surface (Type 24)
Sub-interface

- Interface Type 7 is a general purpose interface and can simulate all types of impact between a set of nodes and a master surface.
- This interface can simulate self-impact, especially buckling during a high speed crash.
- Interface Type 7 solves all problems.
- The main advantage of interface Type 7 is that the stiffness is not constant and increases with penetration preventing the node from going through the shell mid-surface.

접촉조건 설정



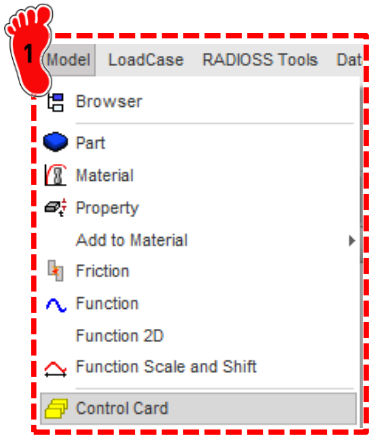
1 LoadCase > Contact Interface클릭,
마우스 우클릭 후 "Multi usage (Type 7) 클릭

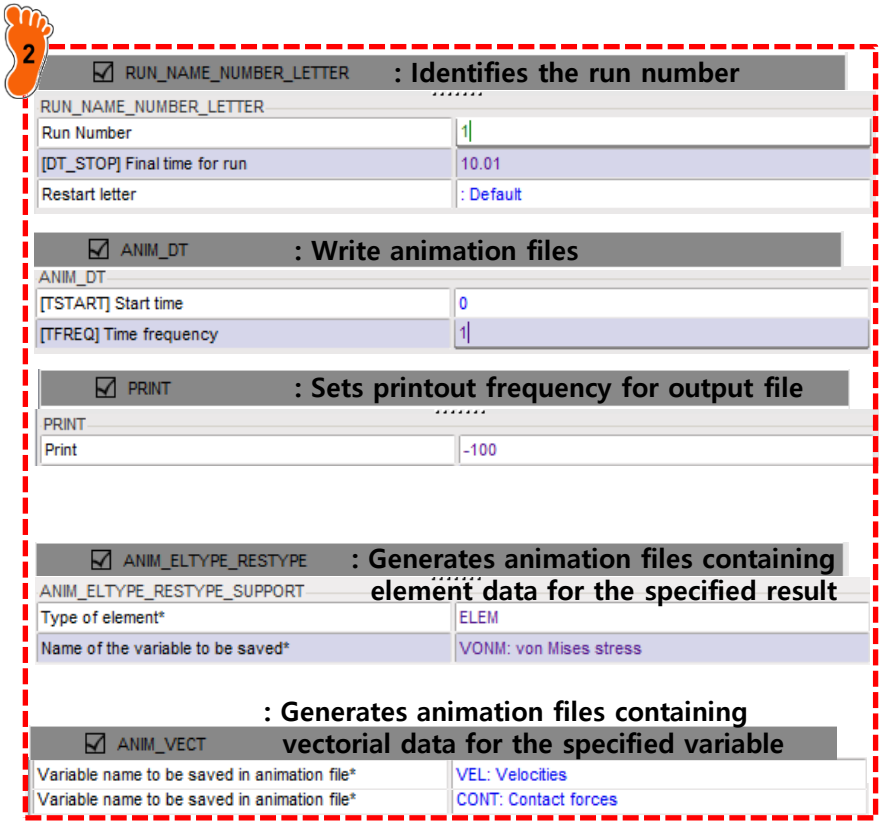
2 Self Impact 체크
[Stfac] : 1
[Gapmin]: 0.9
[Fric]: 0.2
[Mast_id]: 모델 선택

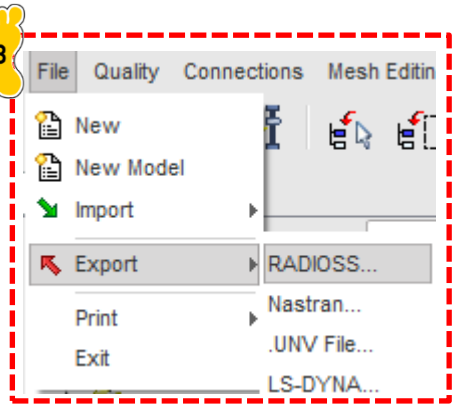
Interface stiffness scale factor
(1.0 real: contact stiffness is taken from the master segment)

Minimum gap for activation of interface

해석 케이스 설정







1 Model > Control Card클릭

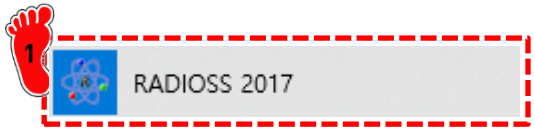
2 파라미터 설정

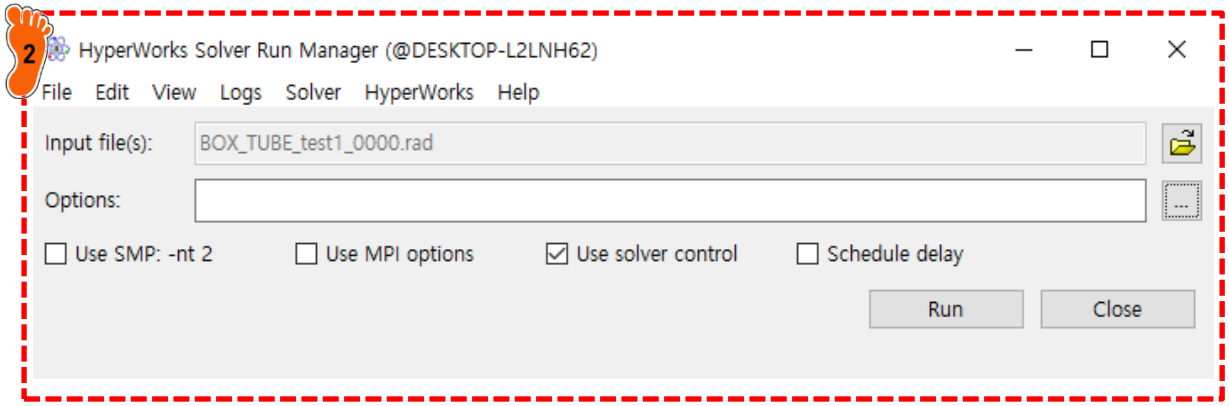
3 모델 export (RADIOSS 선택)

해석 실행 (RADIOSS)

1 Radioss2017 실행

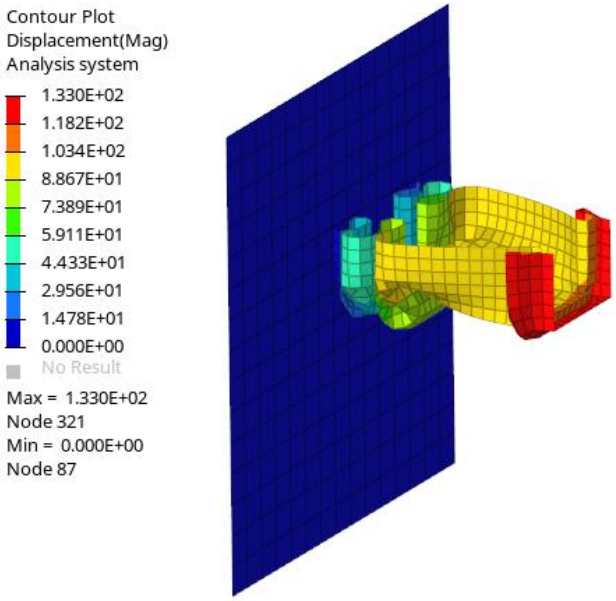
2 앞서 제작한 모델 불러온 뒤 Run 클릭

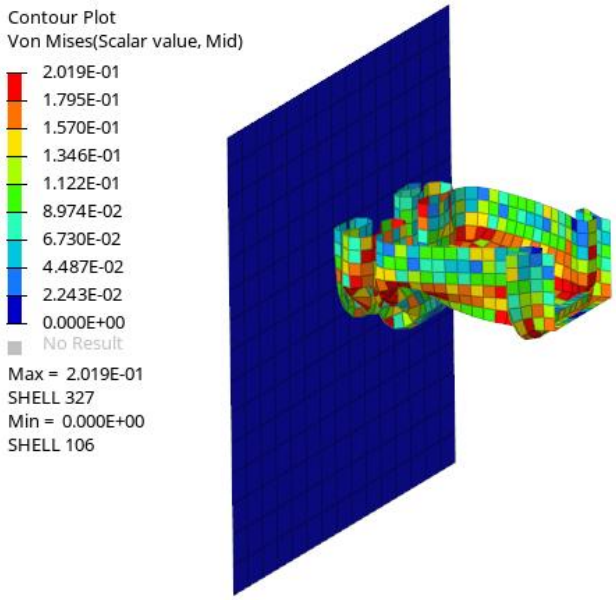




**_0000.rad : Start File-해석 모델
**_0001.rad : 해석 조건 및 결과에서 추출 할 내용 정의

후처리





1 변위, 응력 등 결과 확인

AXIAL CRUSH LOAD OF SQUARE SECTION

예제: BEAM CRUSH ANALYSIS (1)

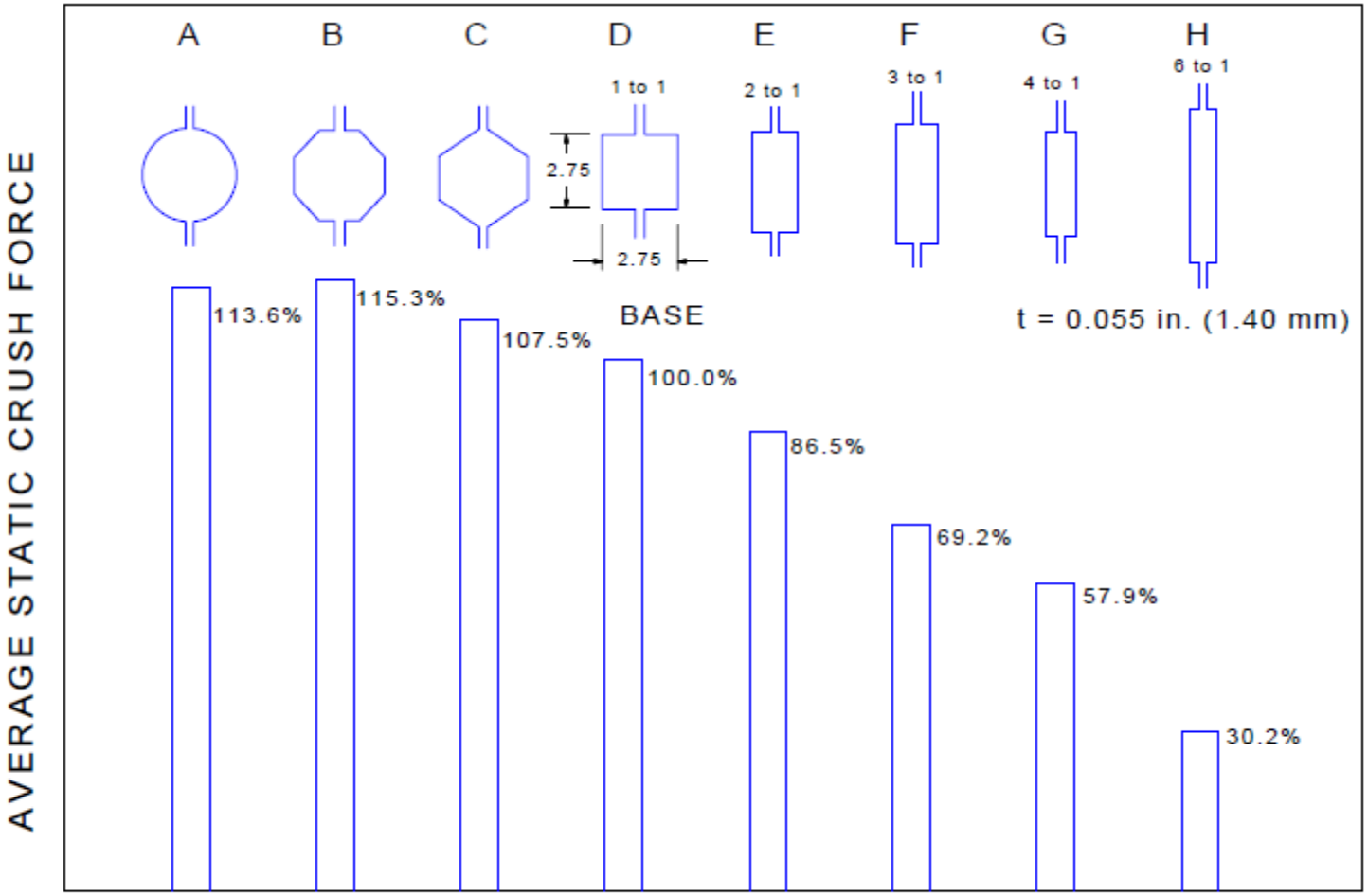
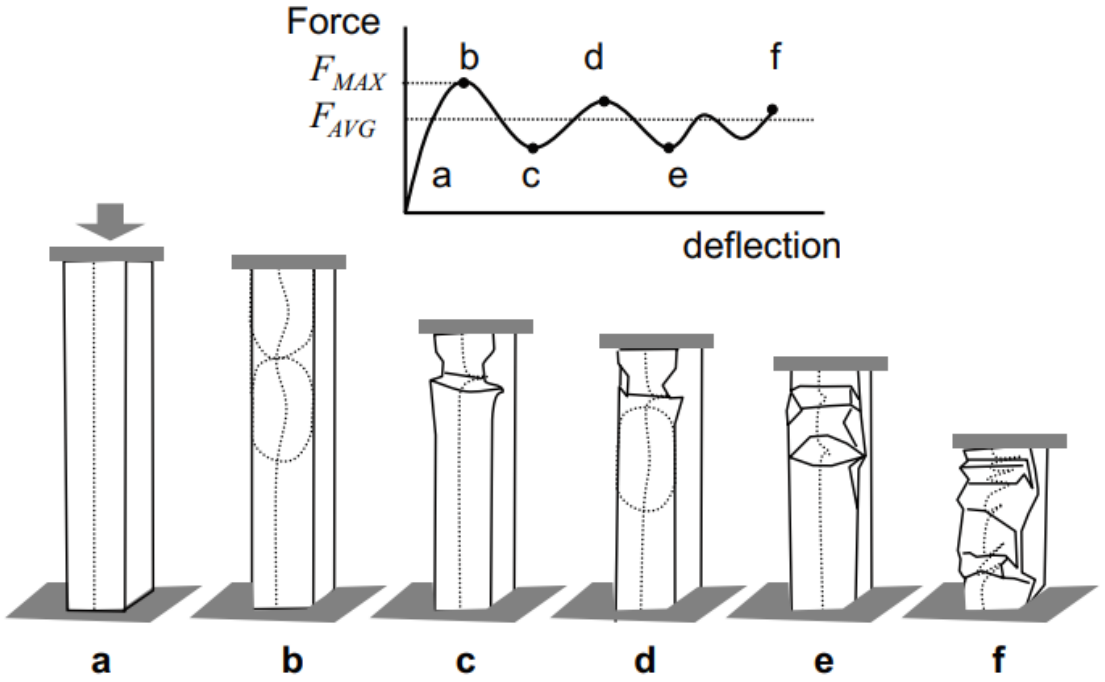


Figure 3.6.4-1 Average static crush force vs. section shape
(All samples were the same mass and length)

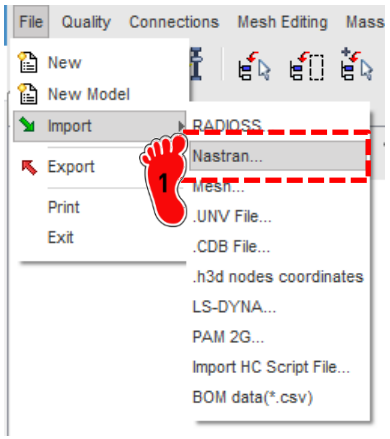
예제: BEAM CRUSH ANALYSIS (2)

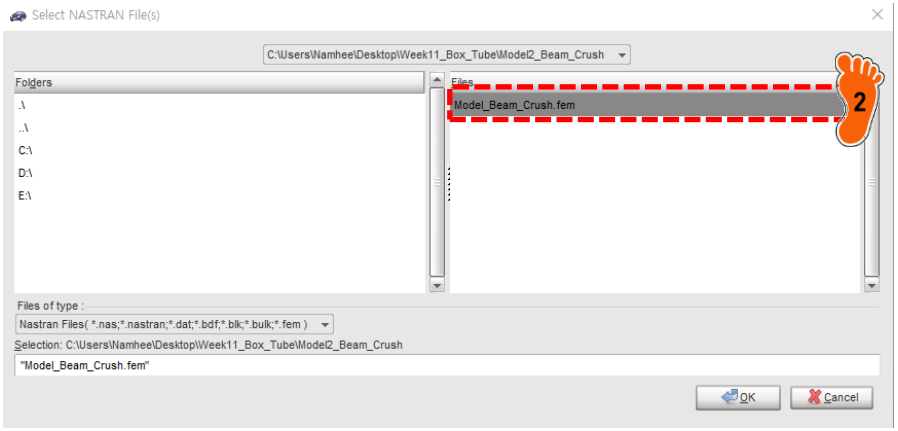
Geometry information

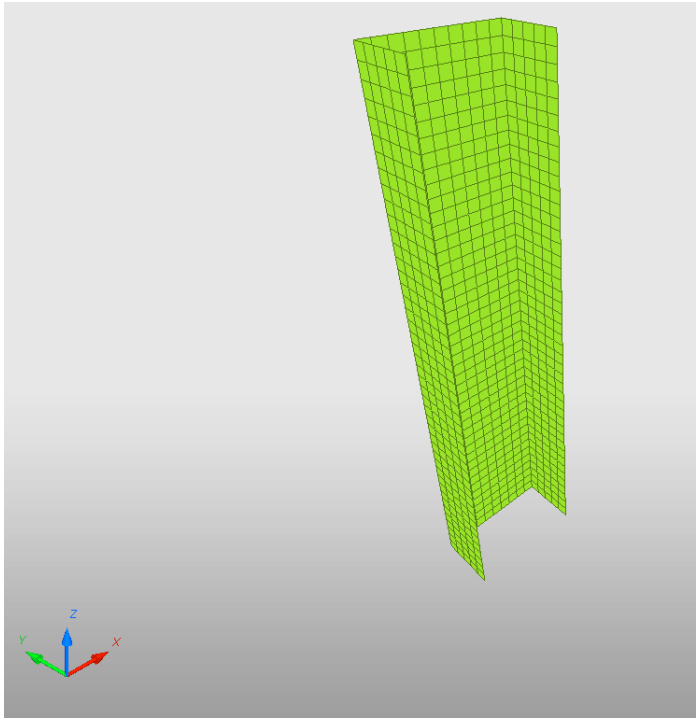
Length: 305 mm
b: 70 mm square
Thickness: 1.4 mm
Yield stress: 247 MPa



유한요소해석 모델 불러오기



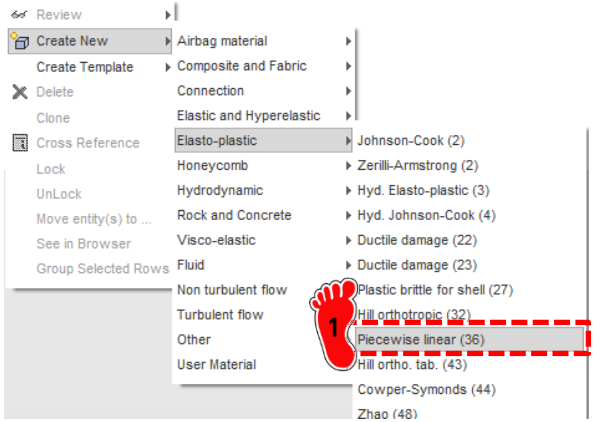




1 File > Import > Nastran 클릭

2 모델 파일 클릭 후 불러오기 (Model_Beam_Crush.fem)

재료 물성 입력 (1)



an isotropic elasto-plastic material using user-defined functions

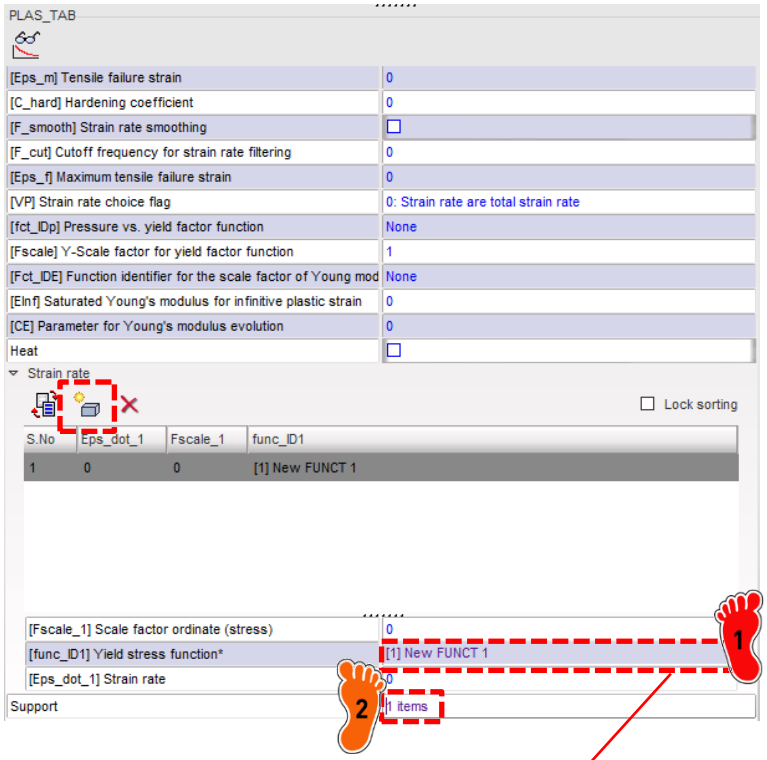
PLAS_TAB

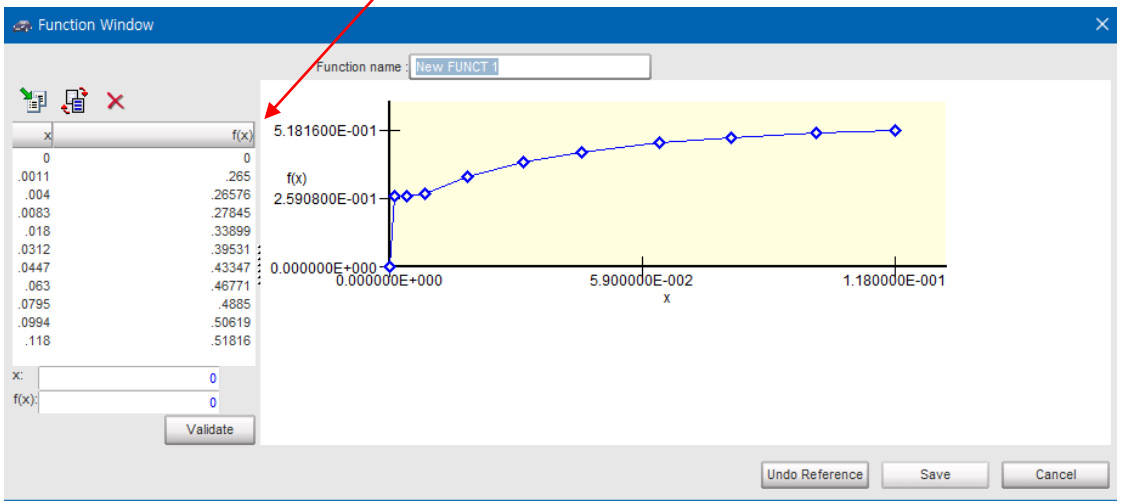
ID	2
Title	New MAT 2
Local Unit System	None
[RHO_I] Initial density*	7.85E-6
[RHO_0] Reference density	0
[E] Young's modulus*	240
[Nu] Poisson's ratio*	.3
[Eps_p_max] Failure plastic strain	0
[Eps_t] Tensile failure strain	0
[Eps_m] Tensile failure strain	0
[C_hard] Hardening coefficient	0
[F_smooth] 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 Model > Material 클릭, 마우스 우클릭 후 Create New > Elasto-plastic > Piecewise linear(36) 재료 생성

2 재료 물성치 입력
[RHO_I]: 7.85e-6 kg/mm^3
[E]: 240 GPa
[Nu]: 0.3

재료 물성 입력 (2)

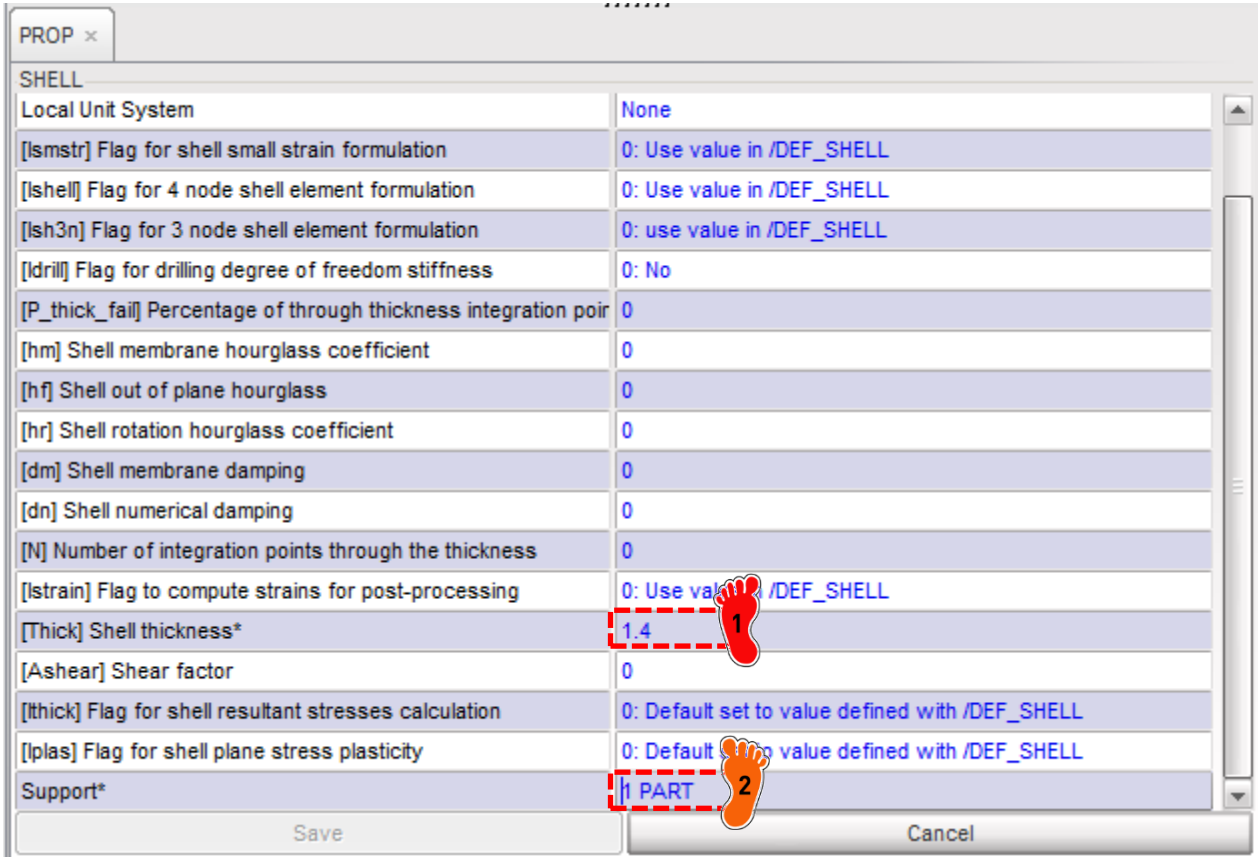




1 Stress-Strain curve 입력

2 모델에 재료 적용

특성 입력

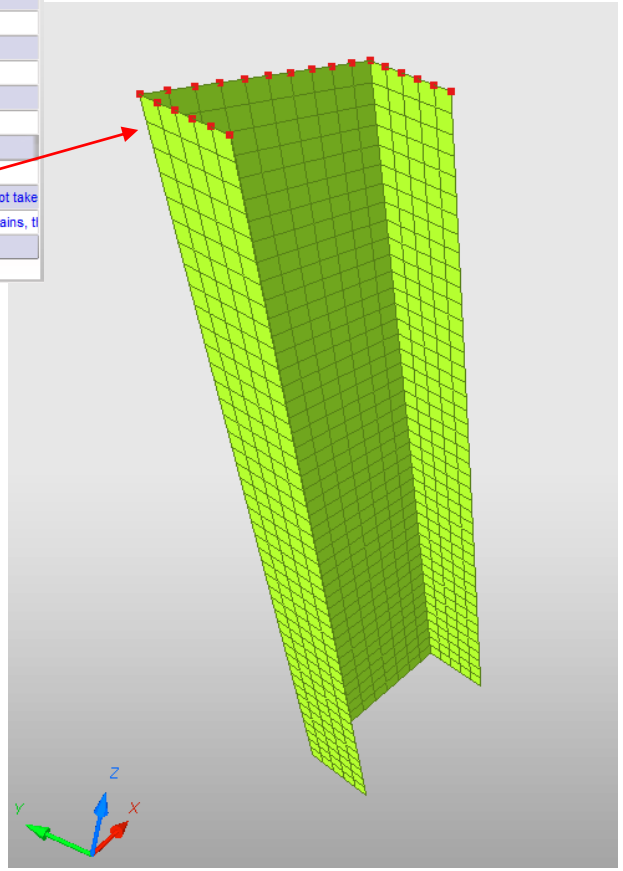


1 Model > Property 클릭, 마우스 우클릭 후 Create New > Surface > Shell (1) 클릭
두께 1.4mm 입력

2 모델에 특성 적용

강체 요소 생성

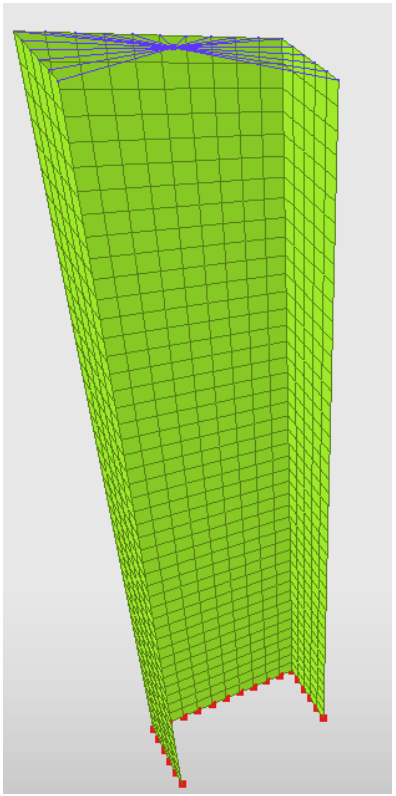
RBODY	
Local Unit System	None
Master node in time History	☐
Lagrange multipliers	☐
Automatic master node	☑
Flag for activate/deactivate in run	0: Not ON/OFF in current Engine file (or docked)
[MASS] Mass	0
[Jxx] JXX inertia	0
[Jyy] JYY inertia	0
[Jzz] JZZ inertia	0
[Jxy] JXY inertia	0
[Jyz] JYZ inertia	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)	

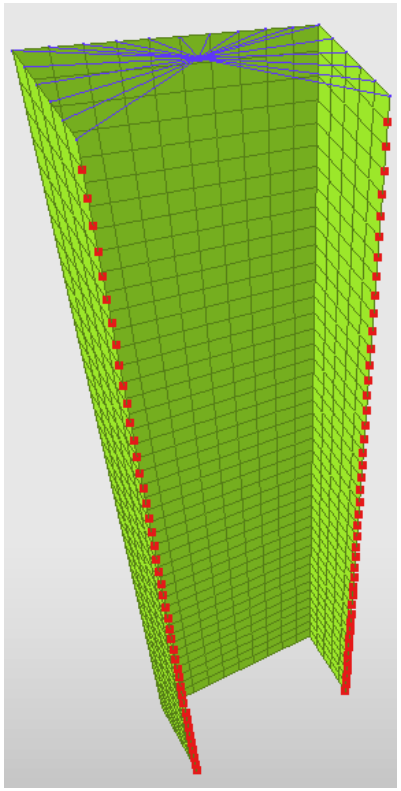


1. Mesh Editing > Rigid Body 클릭, 마우스 우클릭 후 Create New 클릭

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

구속조건 설정 (1)

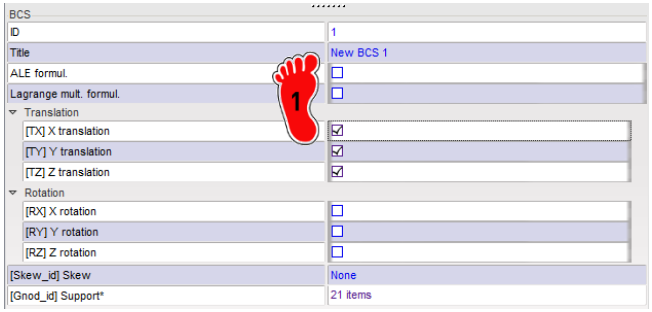


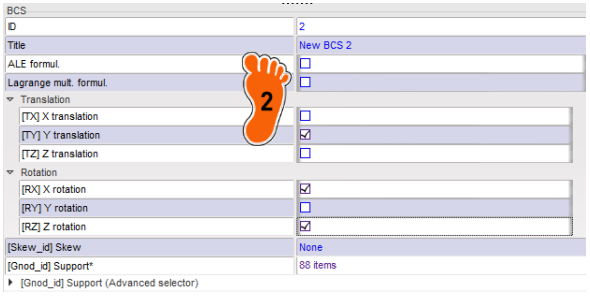


1 LoadCase > Boundary Condition 클릭, 마우스 우클릭 후 Create new 클릭

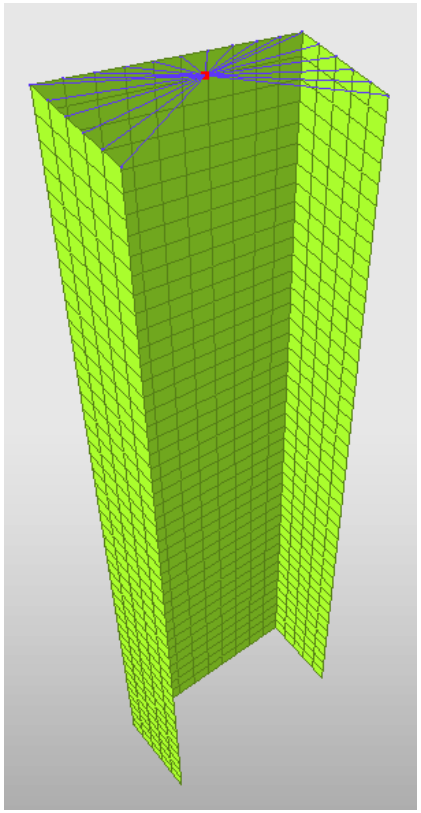
빔의 끝부분에 경계조건 설정 (핀구속 TX, TY, TZ)

2 빔의 잘린 부분은 대칭 조건으로 경계조건 설정 (TY, RX, RZ)





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 Boundary Condition에서 마우스 우클릭 후 Create new 클릭 >

강체의 중앙 부분의 절점은 부딪치는 방향의 자유도를 제외한 모든 자유도 구속

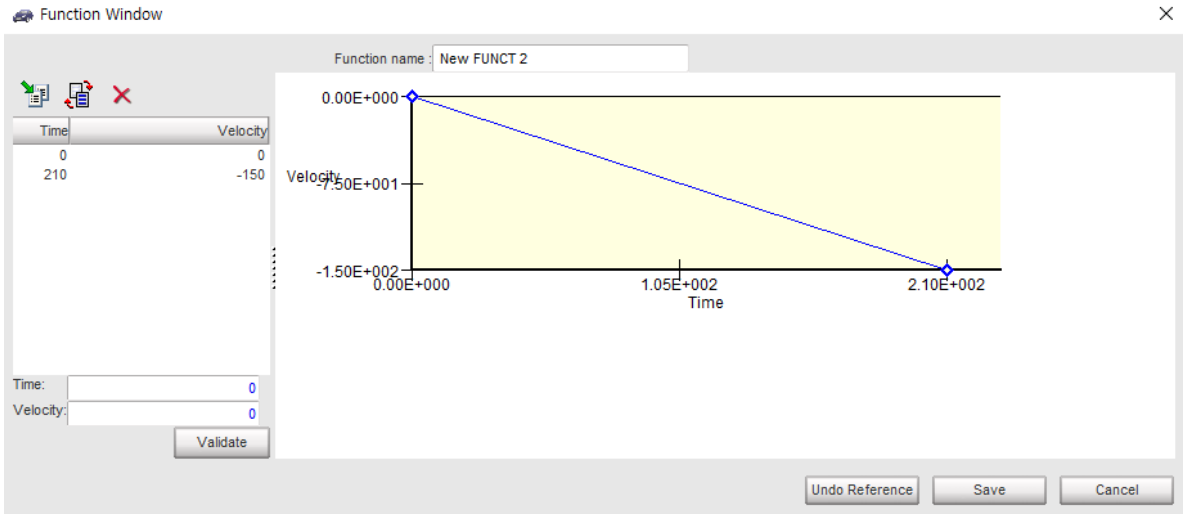
TUBE 변위 설정

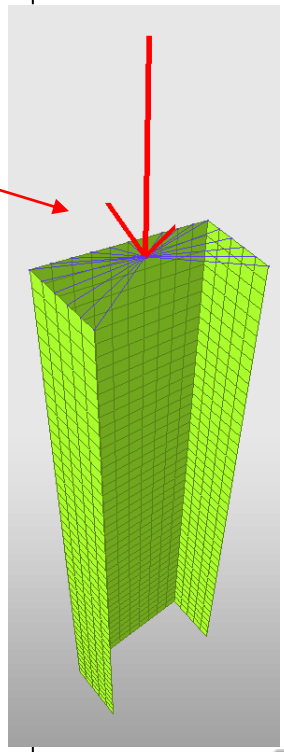
IMPDISP	
ID	1
Title	New IMPDISP 1
Local Unit System	None
[Tstart] Start time	0
[Tstop] Stop time	0
[DIR] Direction*	Z: (translation)
[Ifunct] Time function*	[2] New FUNCT 2
[Scale_x] X-Scale factor	1
[Scale_y] Y-Scale factor	1
[Iskew] Skew	None
[Isensor] Sensor	None
[Frame] Frame	None
[Icoor] Type of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

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

[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 LoadCase > Contact Interface클릭 > 마우스 우클릭 후 > "Multi usage (Type 7) 클릭 (contact interface 생성)

2 파라미터 설정
Self Impact: 체크
[Stfac]: 1
[Gapmin]: 0.9
[Fric]: 0.2

접촉 파트 설정 (모델 선택)

해석 케이스 설정

1 Model > Control card 설정

파라미터 설정

모델 export (RADIOSS 선택)

☑ RUN_NAME_NUMBER_LETTER : Identifies the run number

RUN_NAME_NUMBER_LETTER

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

☑ ANIM_DT : Write animation files

ANIM_DT

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

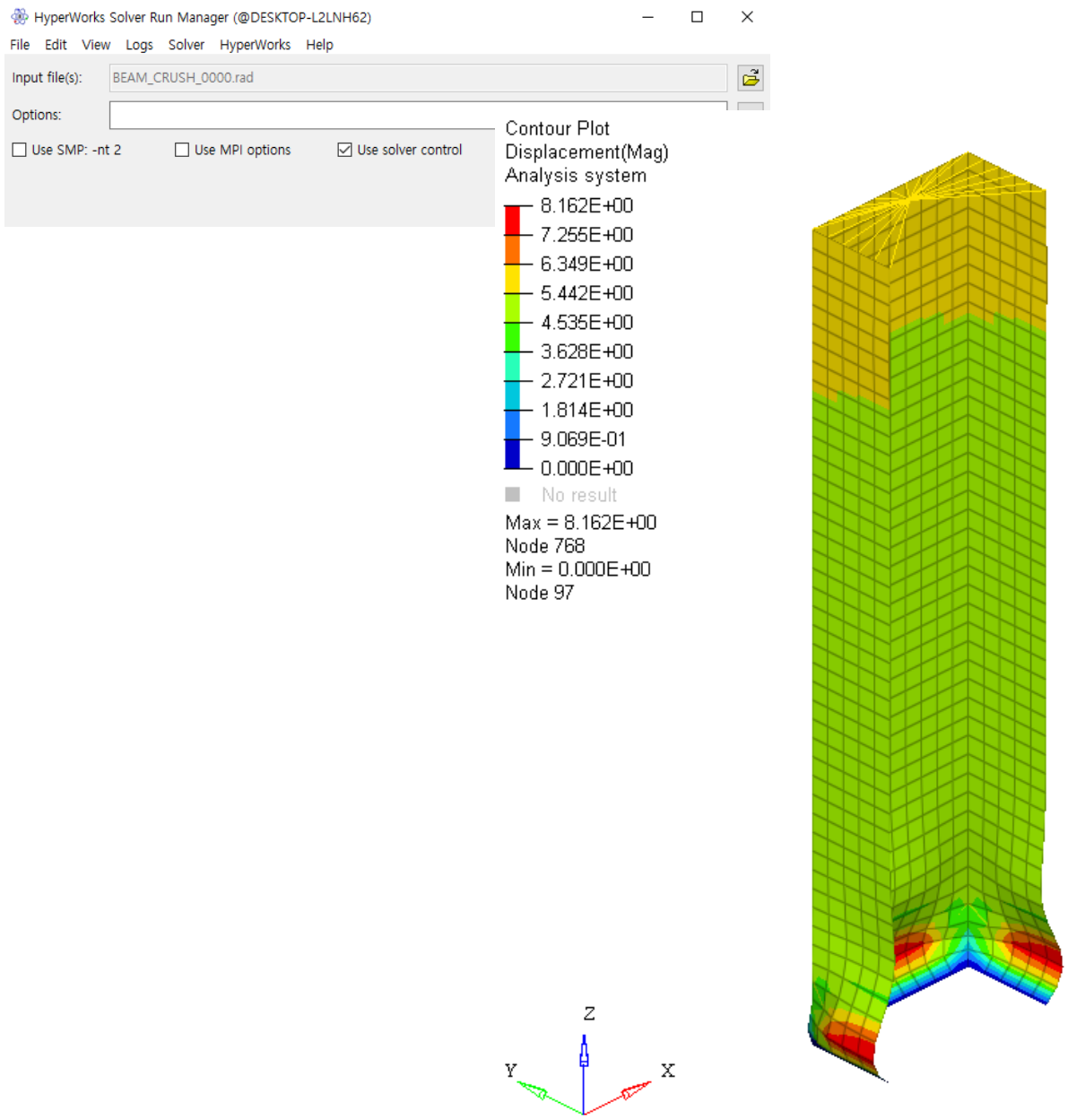
☑ ANIM_VECT : Generates animation files containing vectorial data for the specified variable

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

ANIM_VECT_RESTYPE_SUPPORT

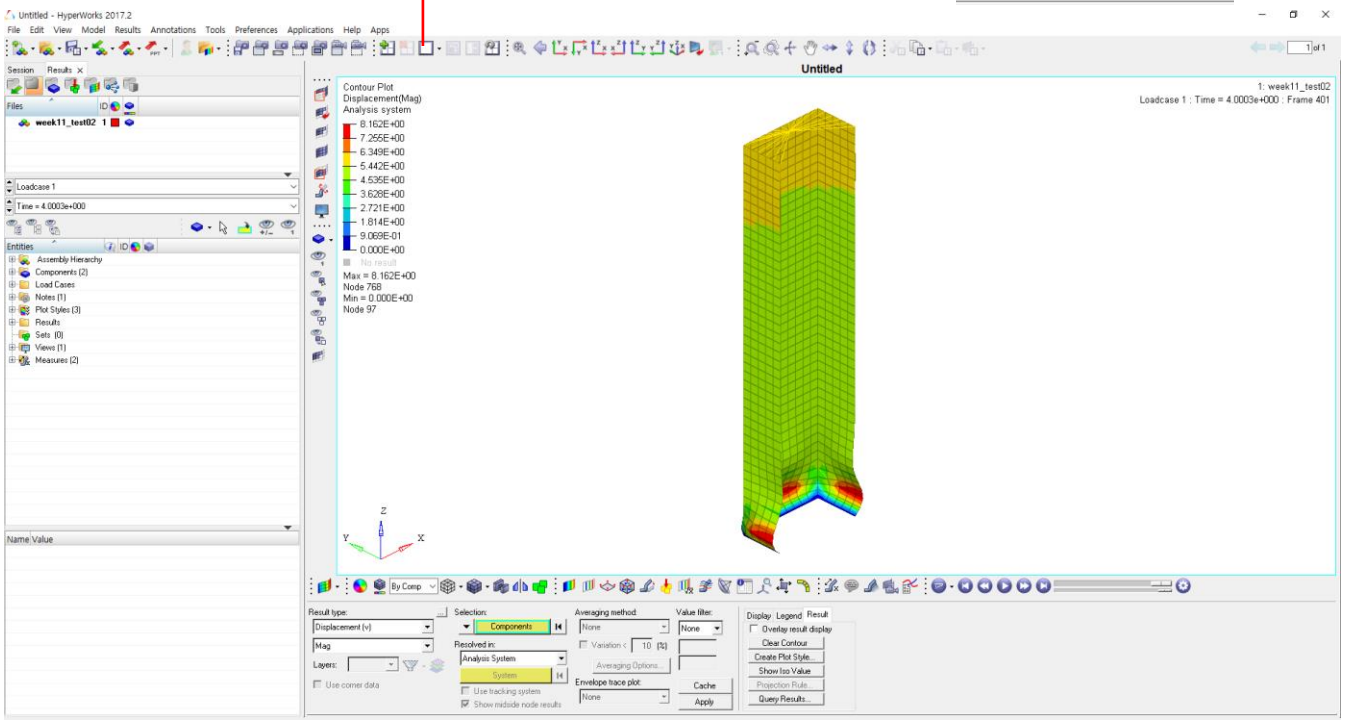
Variable name to be saved in animation file*	FOPT: Forces and moments for rigid bodies, rigid walls and sect

후처리 (1)



1 RADIOSS 에서 해석 실행

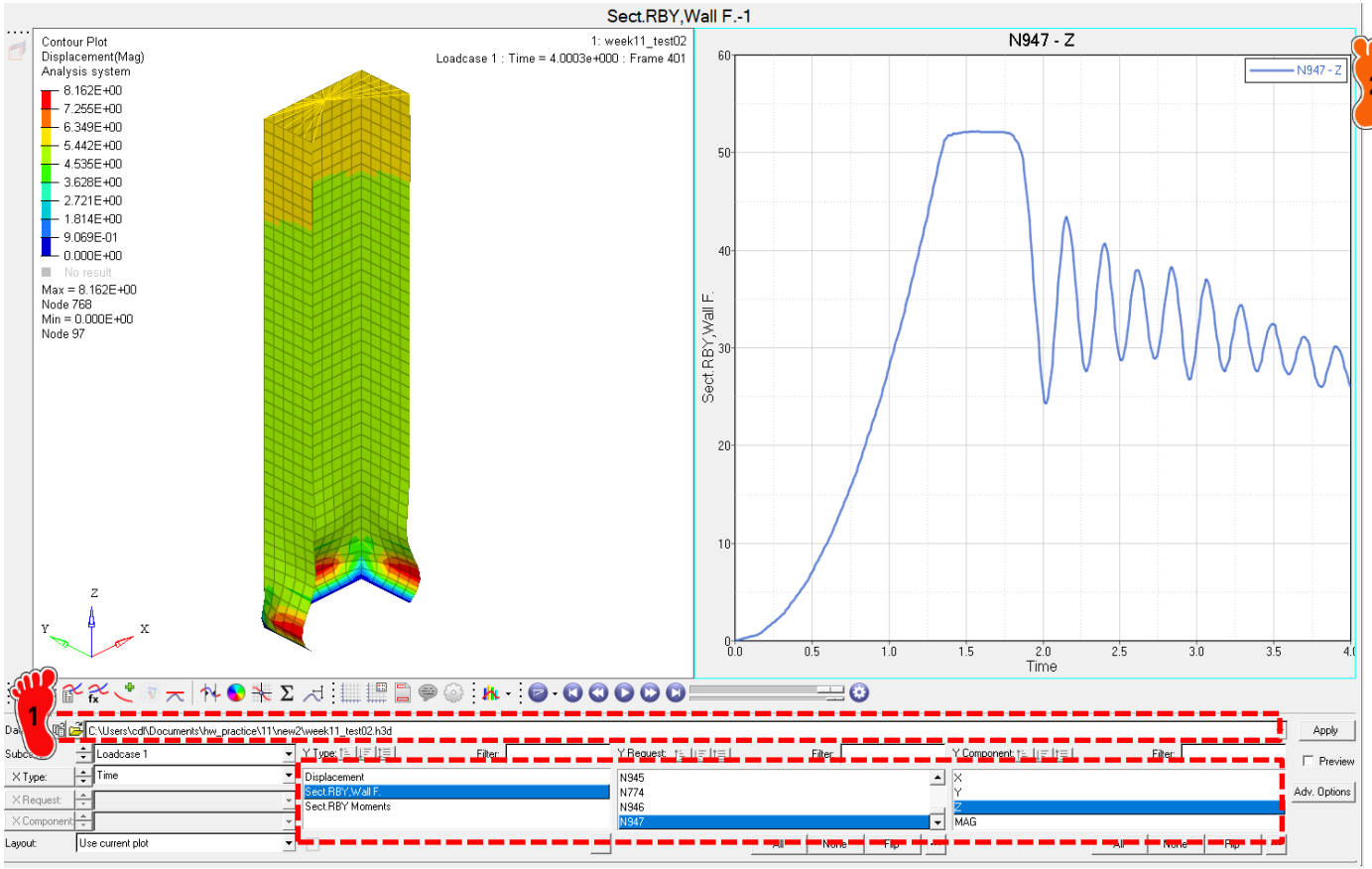
후처리 (2)



1 HyperView에서 Select from list 버튼 클릭 > 화면 분할

분할된 화면 선택 > HyperGraph 2D 선택

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

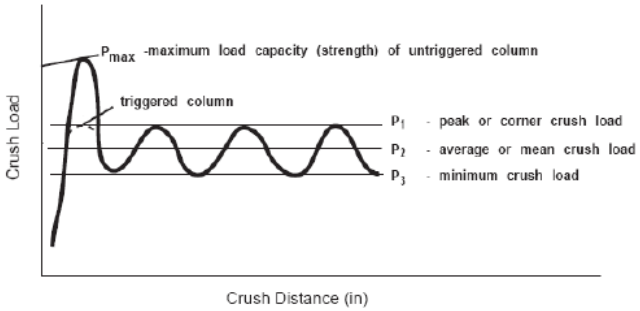
$$P_M = 386t^{1.86}b^{0.14}\sigma_Y^{0.57} = 30237 \text{ N}$$

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

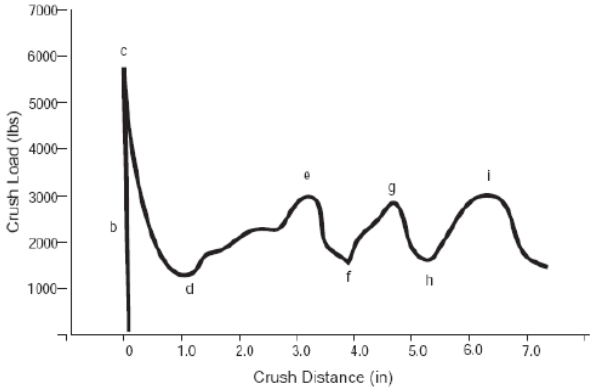
$$P_1 = 1.42P_M = 42936 \text{ N}$$

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

Ideal graph

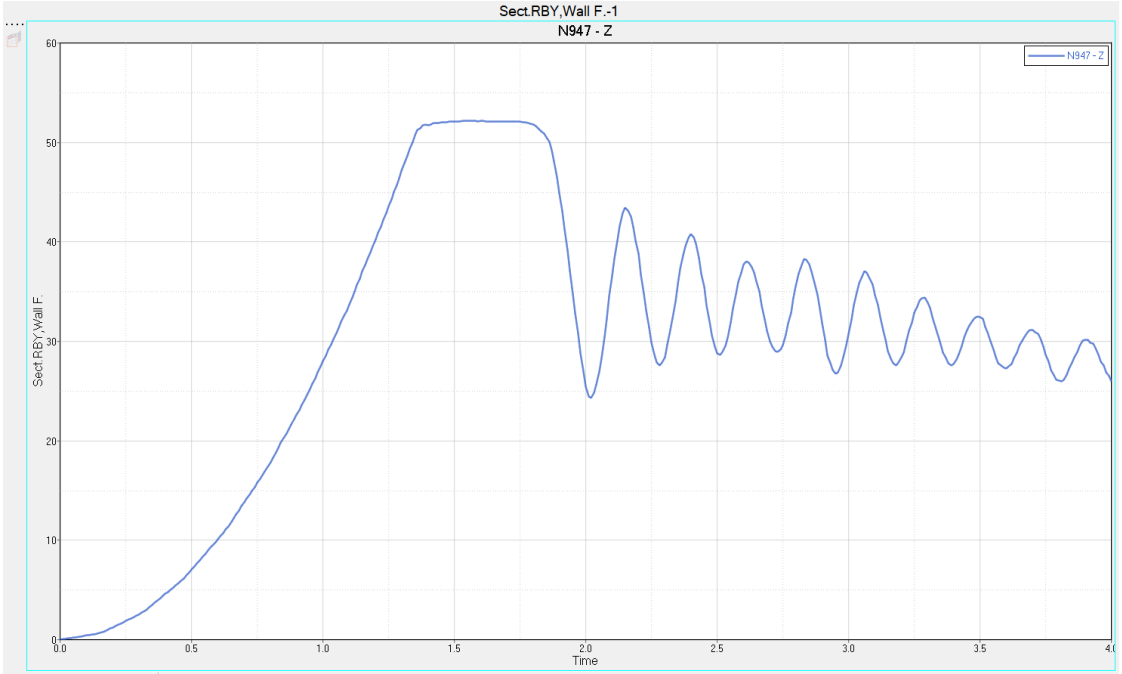


Practical graph



FEM solution

$$P_{MAX} = 52145 \text{ N}$$



숙제

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

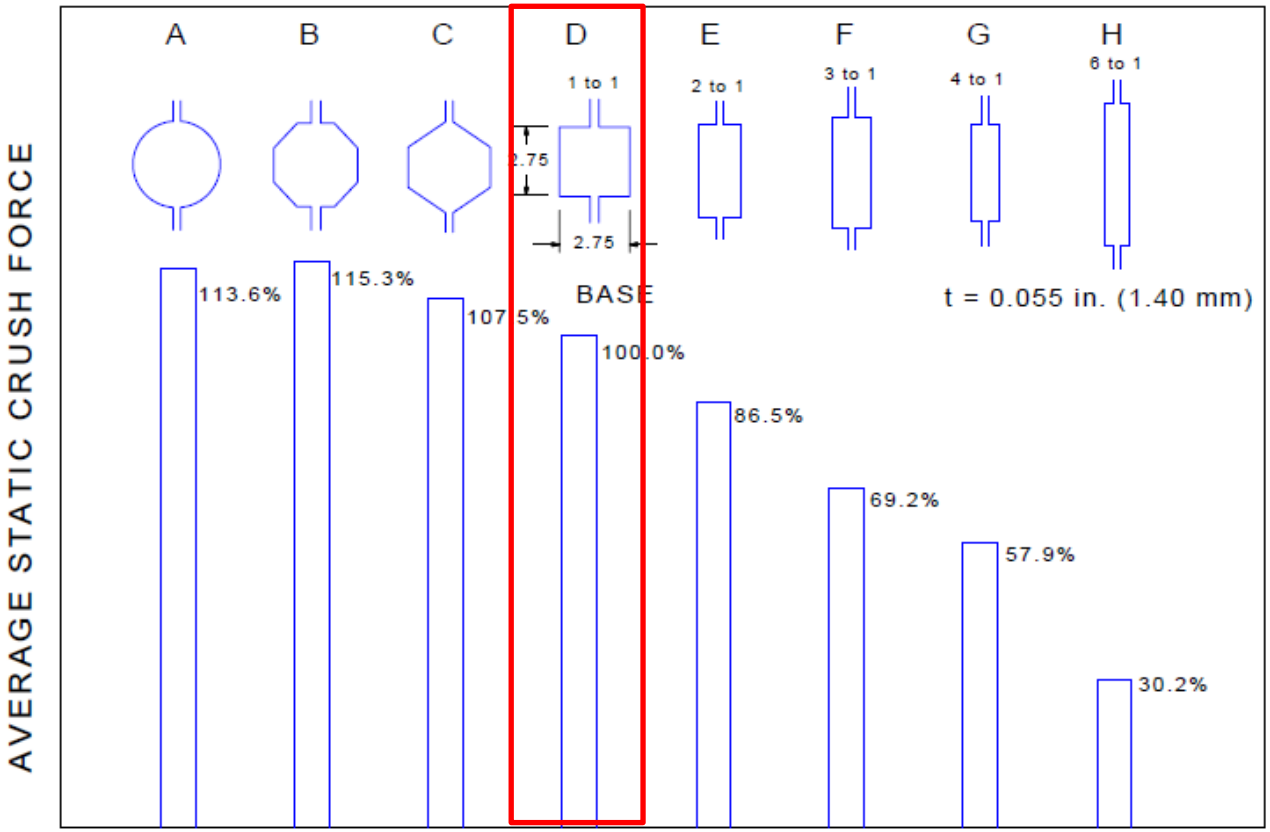


Figure 3.6.4-1 Average static crush force vs. section shape
(All samples were the same mass and length)