coreform

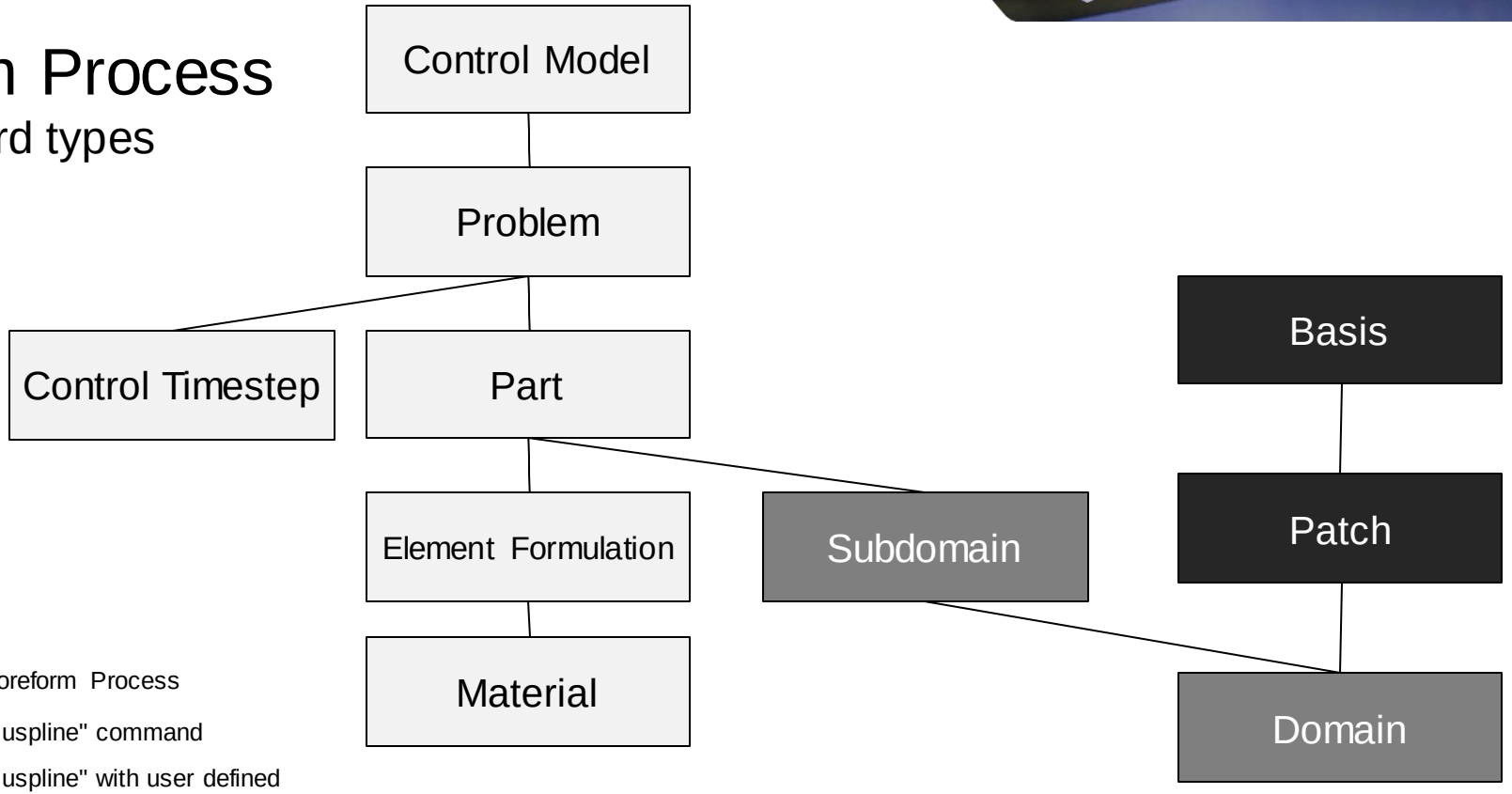
Better simulation through better geometry

Short Course Example Problems

February 20, 2020
Provo, UT

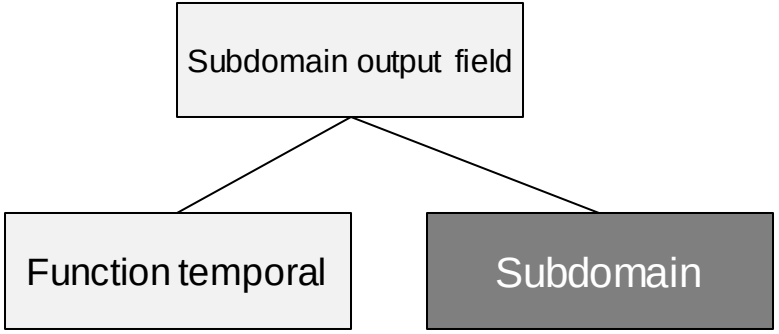
Coreform Process

Required card types



Coreform Process

Output cards



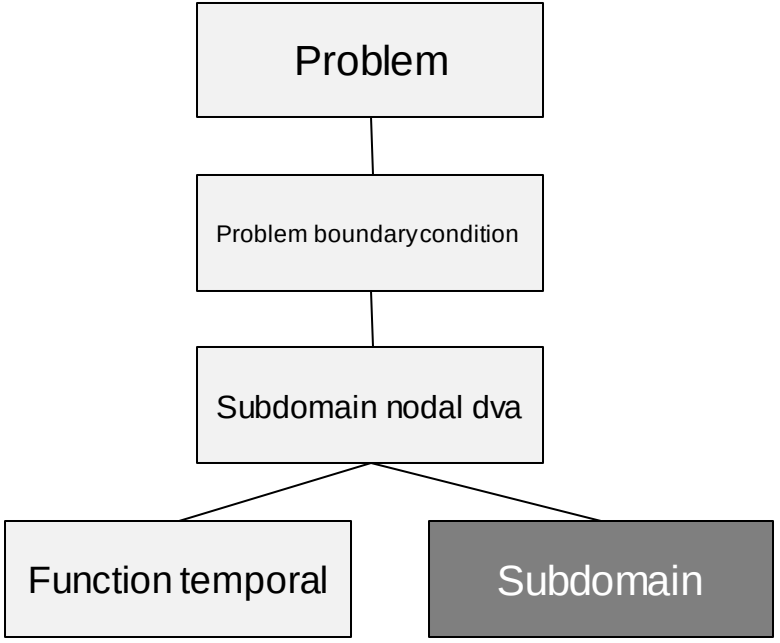
User-defined in Coreform Process

Created by "build uspline" command

Created by "build uspline" with user defined setting

Coreform Process

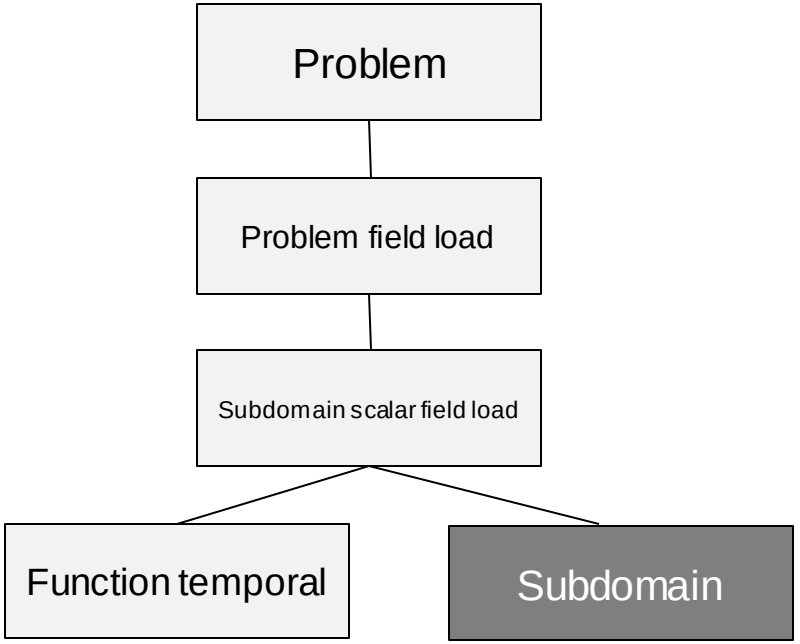
Displacement Boundary Condition cards



- User-defined in Coreform Process
- Created by "build uspline" command
- Created by "build uspline" with user defined setting

Coreform Process

Pressure load cards



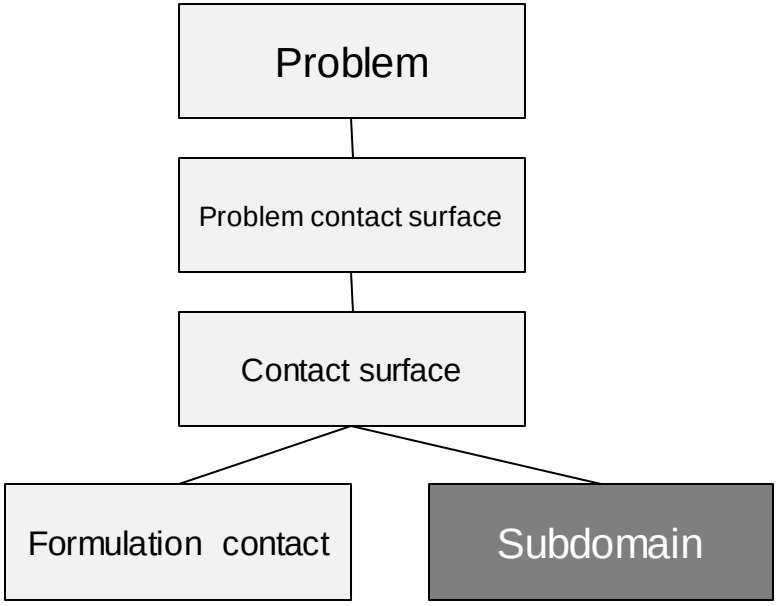
User-defined in Coreform Process

Created by "build uspline" command

Created by "build uspline" with user defined setting

Coreform Process

Contact cards



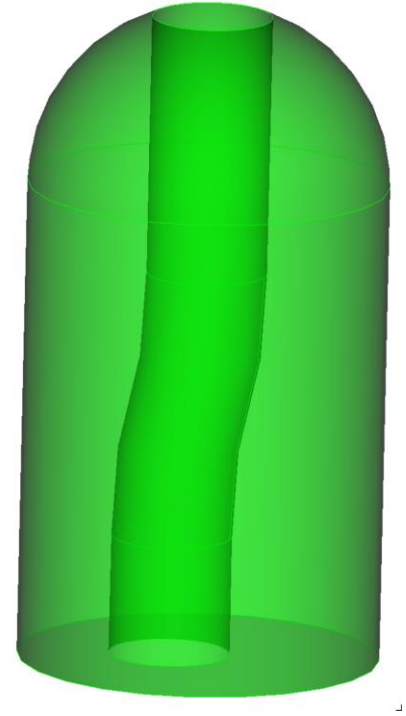
User-defined in Coreform Process

Created by "build uspline" command

Created by "build uspline" with user defined setting

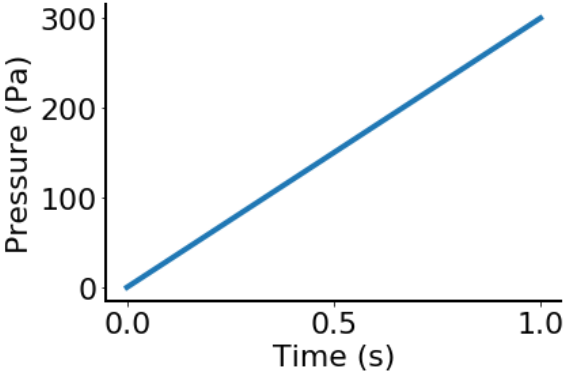
Problem 1: Simple unstructured solid simulation

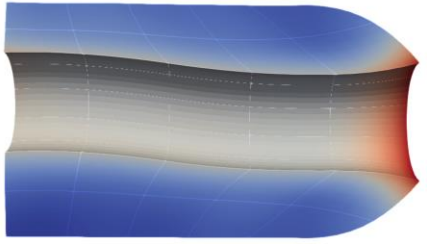
Overview



This tutorial will simulate the dynamic pressure loading of the part shown on the left. The pressure graphed on the right will be applied to the inside of the part, and the resulting displacement will be observed.

The problem will be modelled using half of the given part, with symmetry boundary conditions.



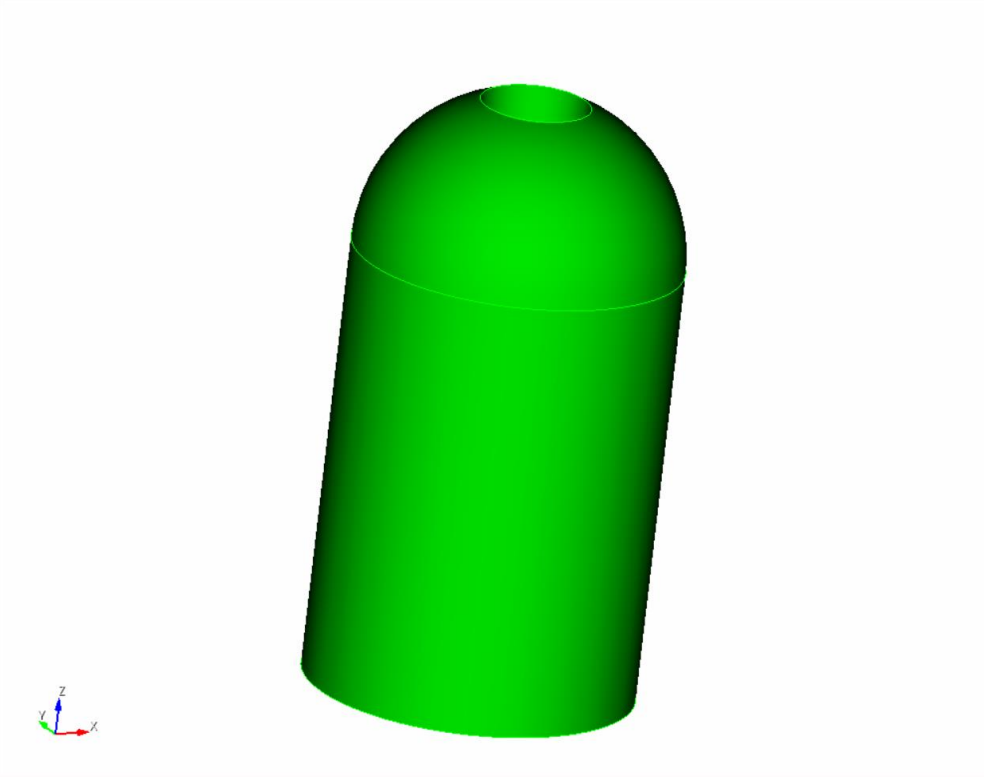


Learning Objectives

Tutorial participants will:

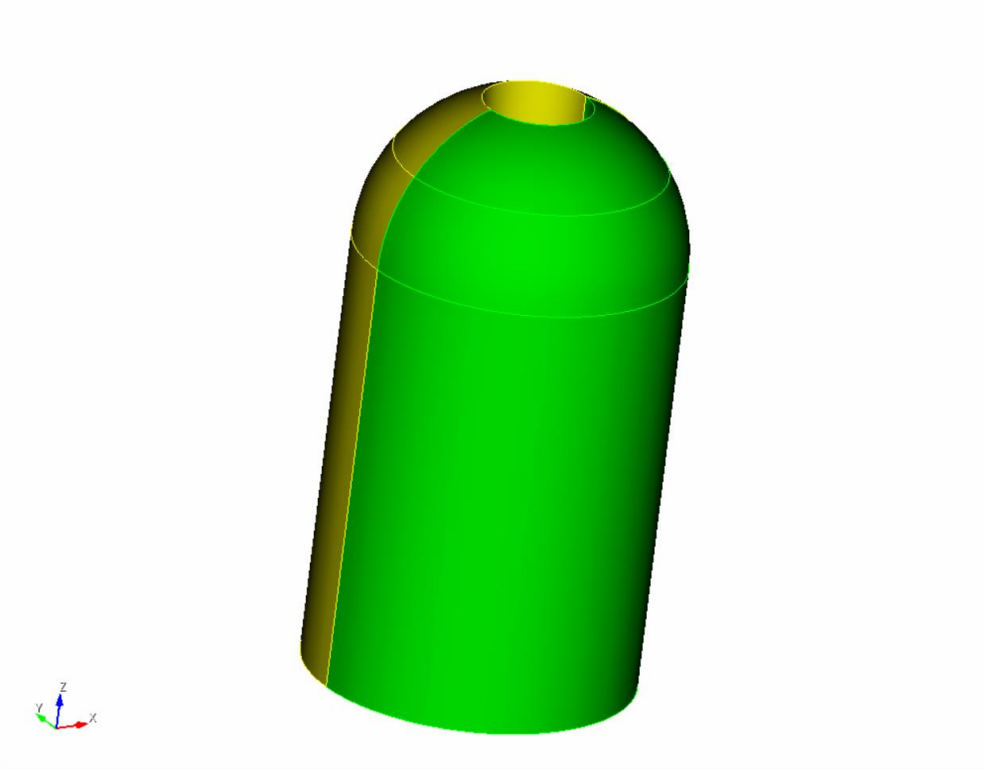
- Mesh a swept solid in Trelis and build a smooth spline over the mesh
- Set up and run a simple simulation on a semi-unstructured solid

Open Part



This file is available at https://coreform.com/shortcourse/PressureVessel/pressurevesselpart.trelis

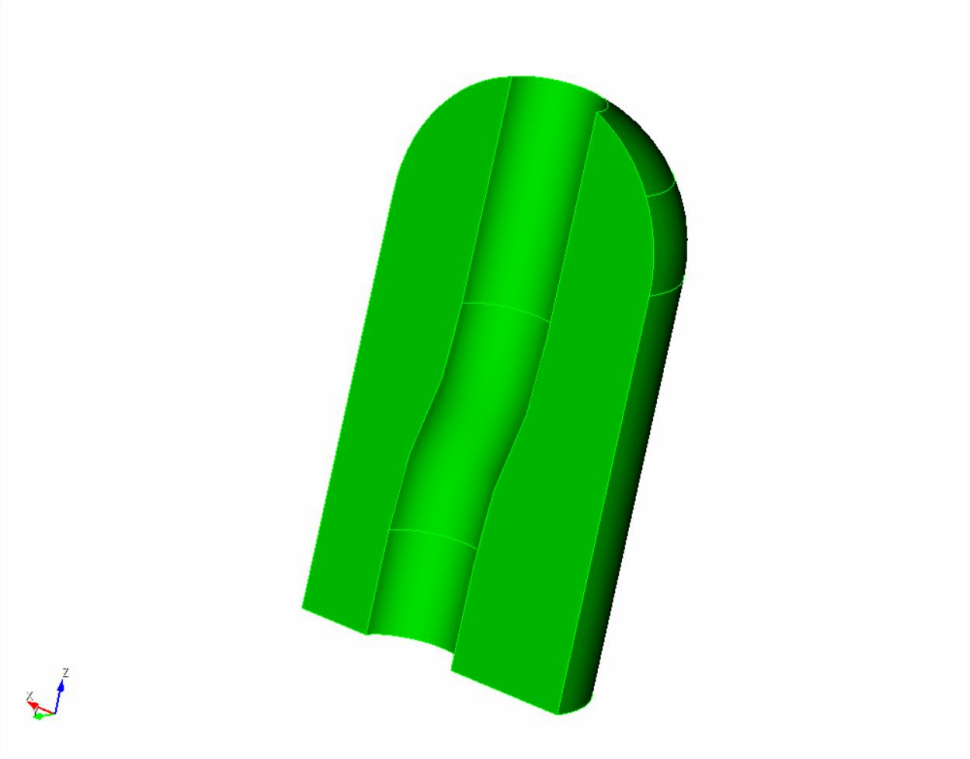
Webcut part on ZX plane



- or -

```
webcut volume 1 with plane yplane
```

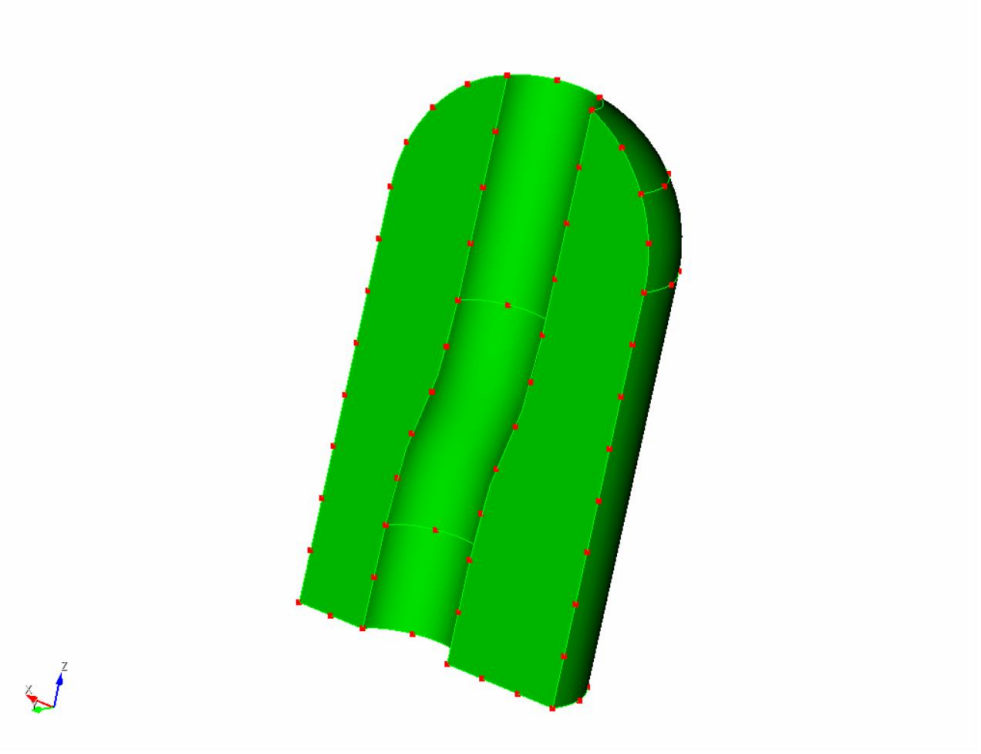
Delete half of the part

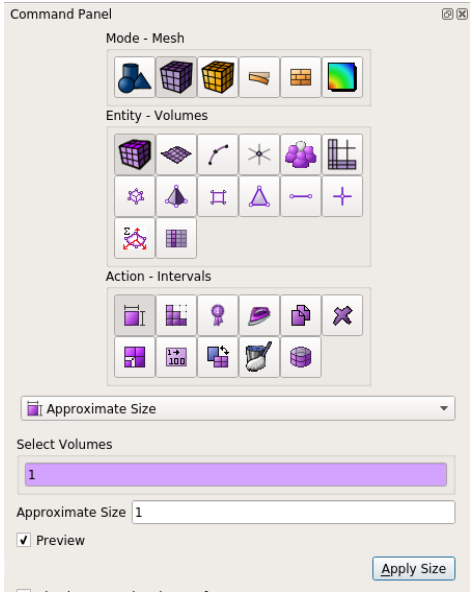


- or -

```
delete volume 2
```

Set mesh element size

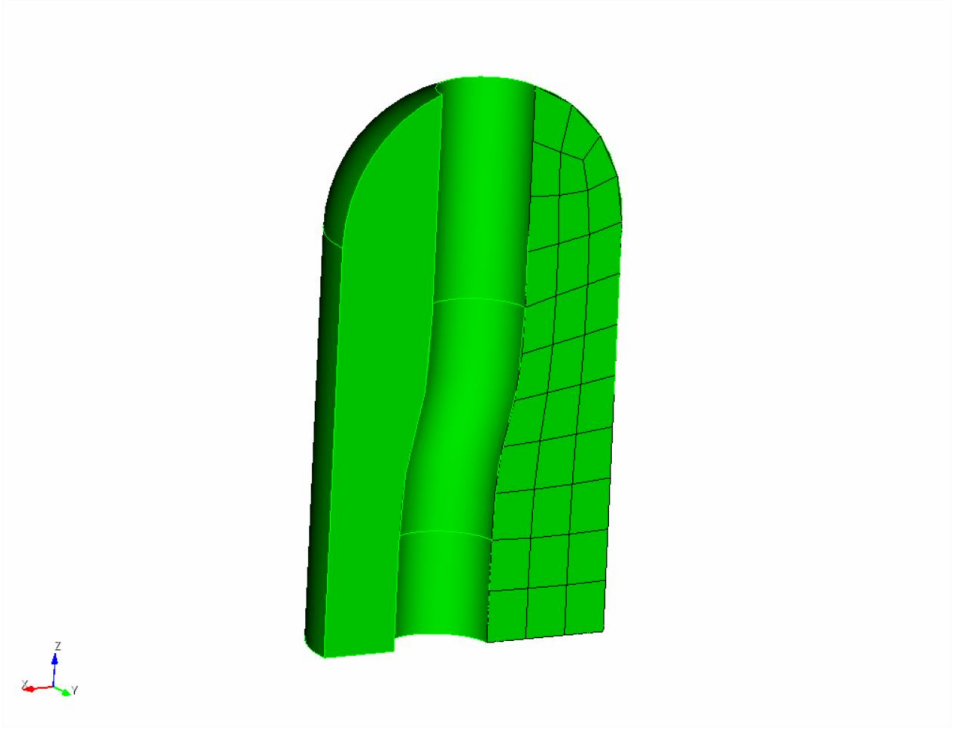


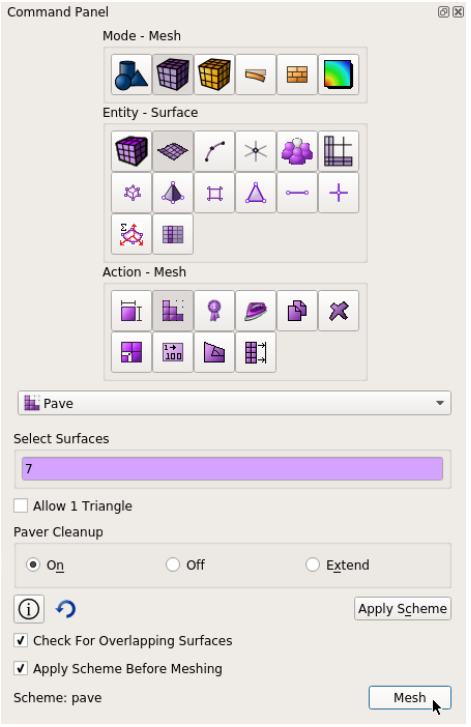


- or -

```
volume 1 size 1
```

Mesh face

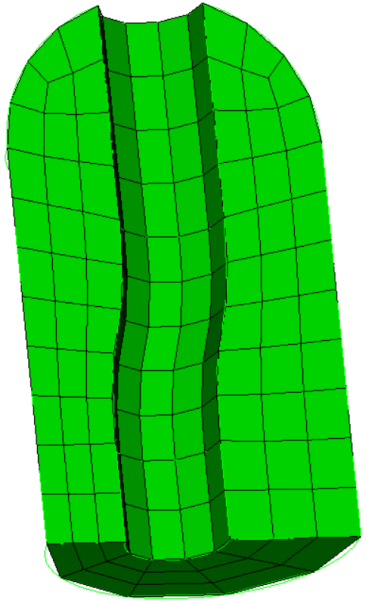


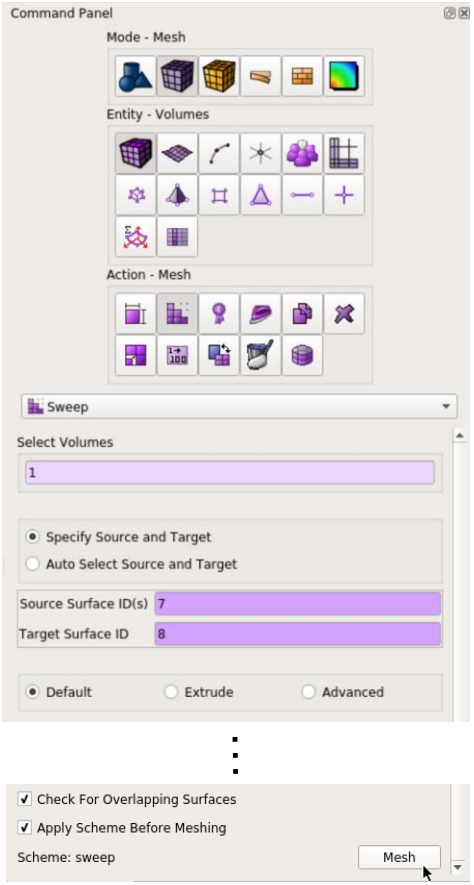


- or -

```
surface 7 scheme pave
mesh surface 7
```

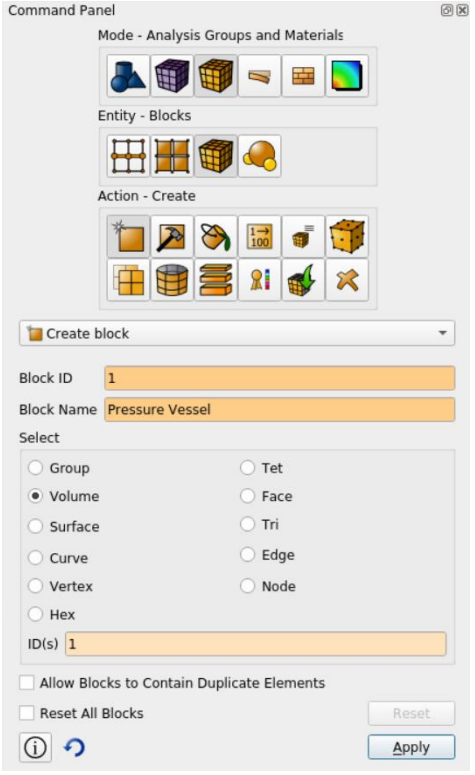
Mesh volume as swept surface






```
volume 1 scheme sweep source surface 7 target surface 8
mesh volume 1
```

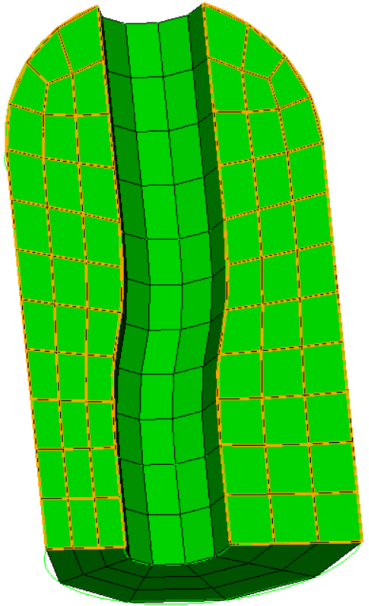
Create block



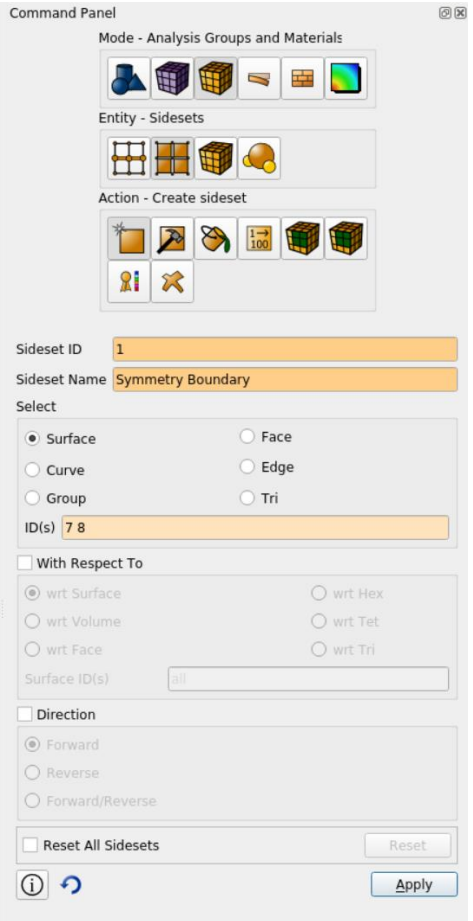
- or -

```
block 1 add volume 1
block 1 name "Pressure Vessel"
```

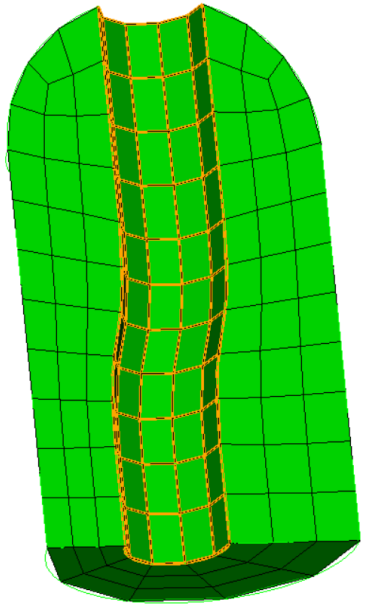
Create sidesets – symmetry boundary



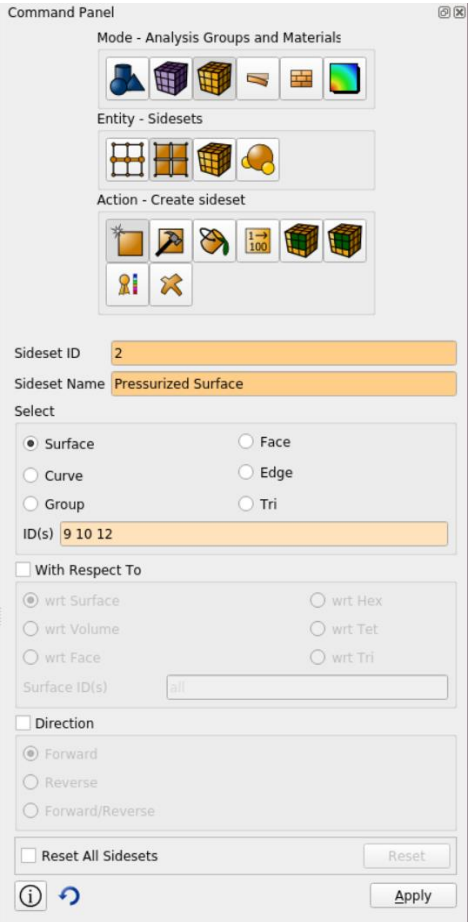
```
sideset 1 add surface 7 8
sideset 1 name "Symmetry Boundary"
```



Create sidesets – pressurized surface

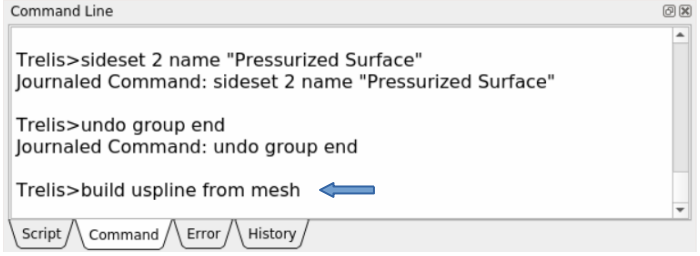


```
sideset 2 add surface 9 10 12
sideset 2 name "Pressurized Surface"
```



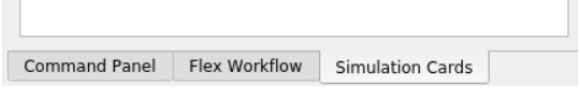
The mesh at this point is available at
https://coreform.com/shortcourse/PressureVessel/pressurevesselmesh.trelis

Generate and view simulation geometry cards

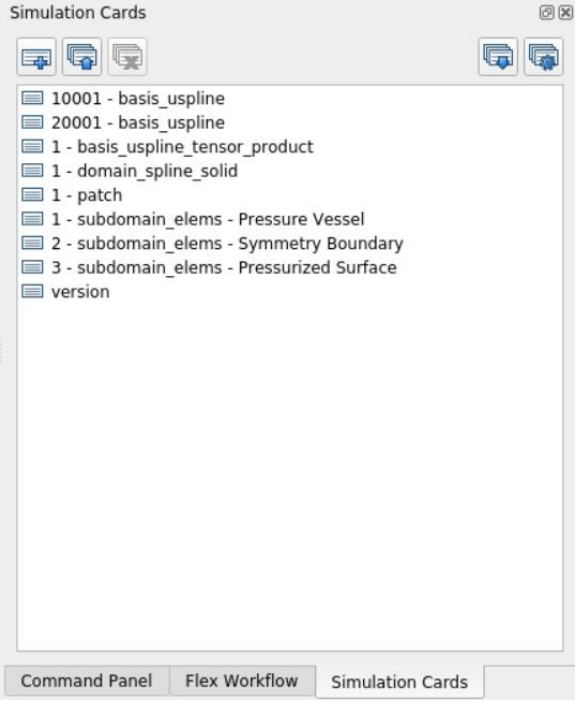


Build is finished when Trelis> prompt returns.

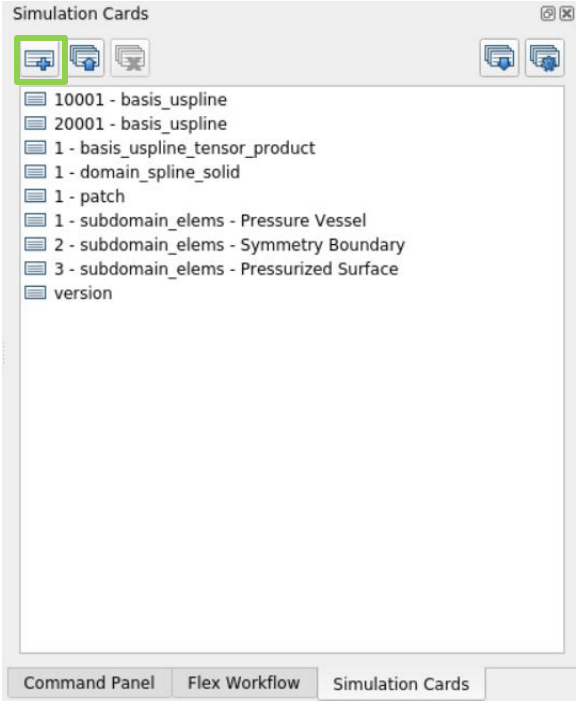
Select "Simulation Cards" tab in bottom right

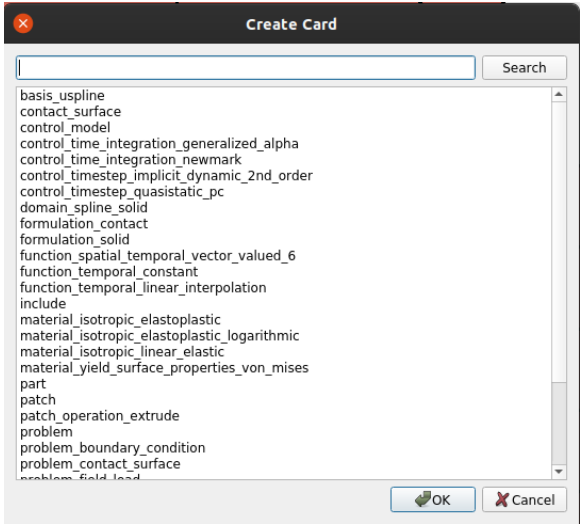


The simulation cards tab should now look like this:

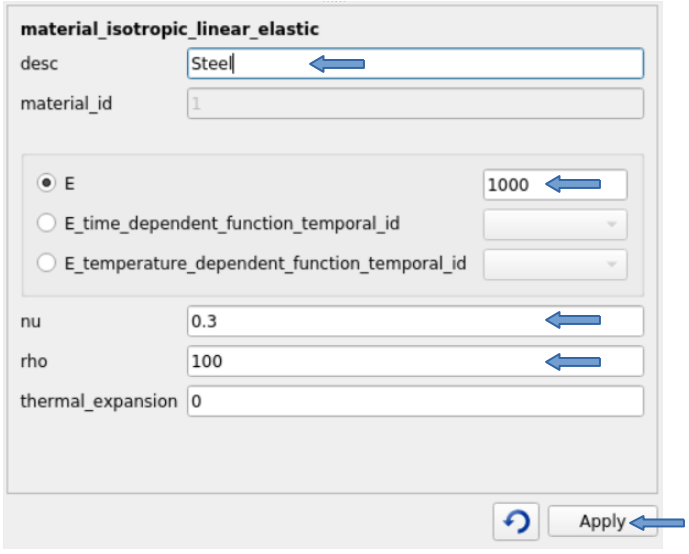


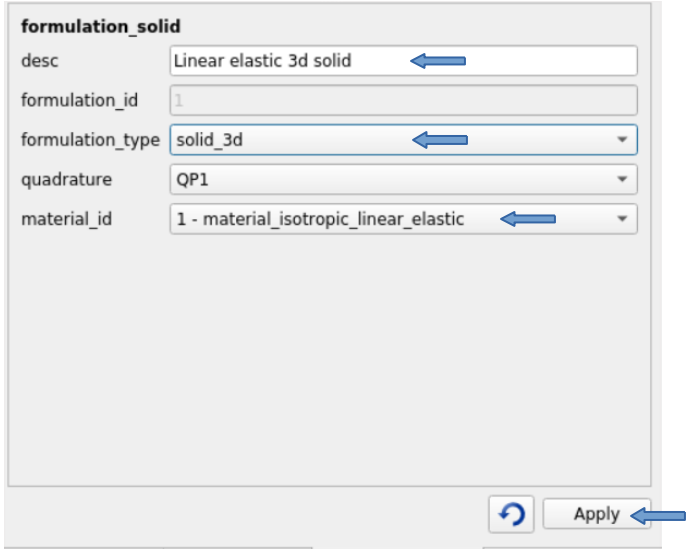
Adding Cards



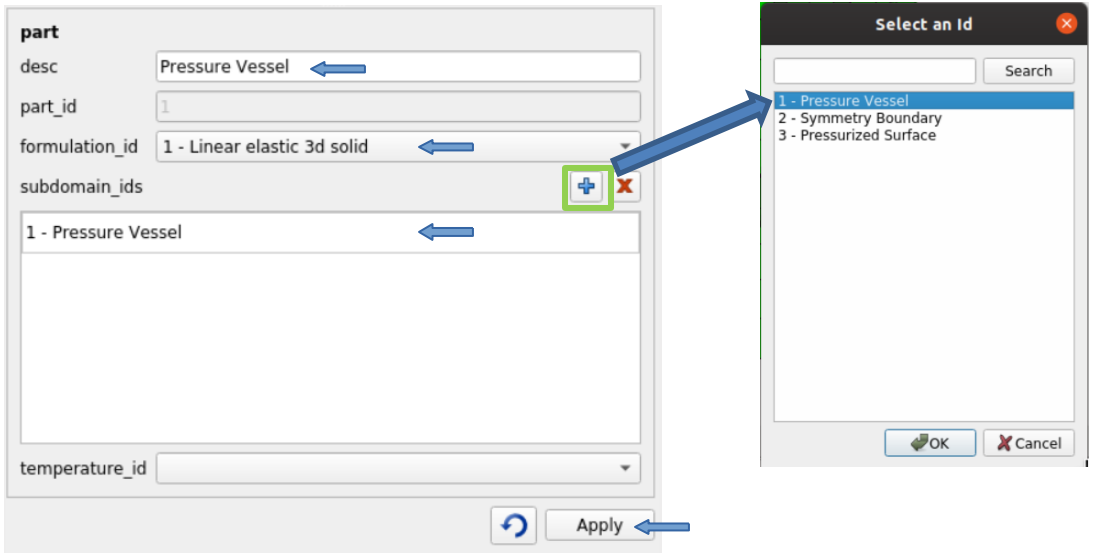


Add material and formulation cards

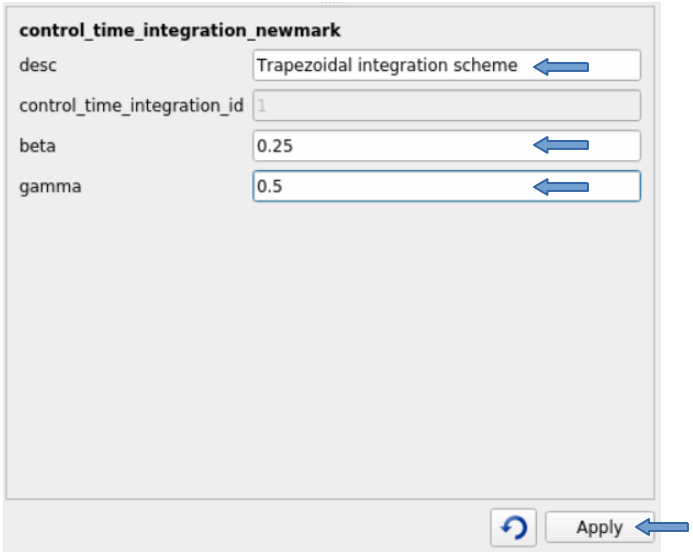


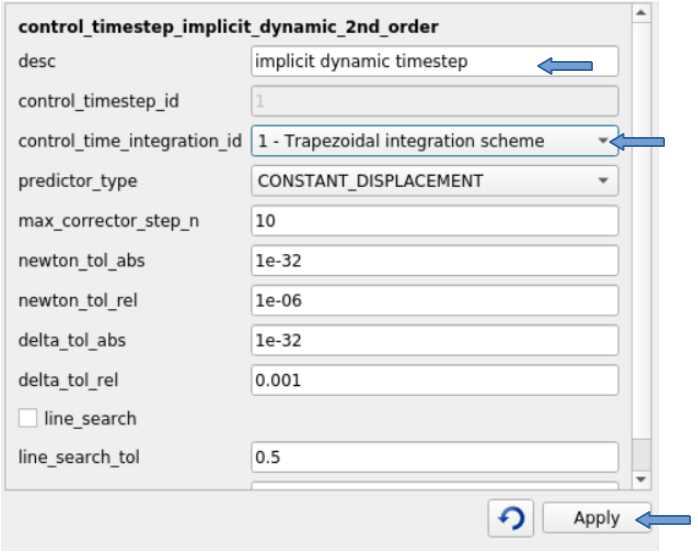


Add part card

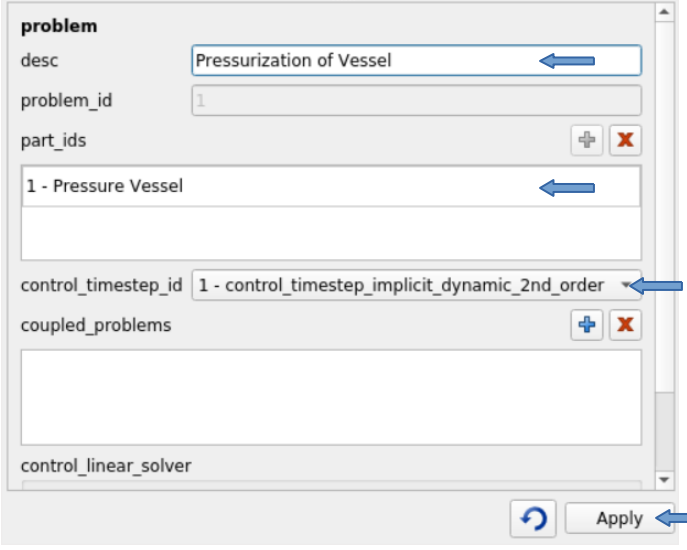


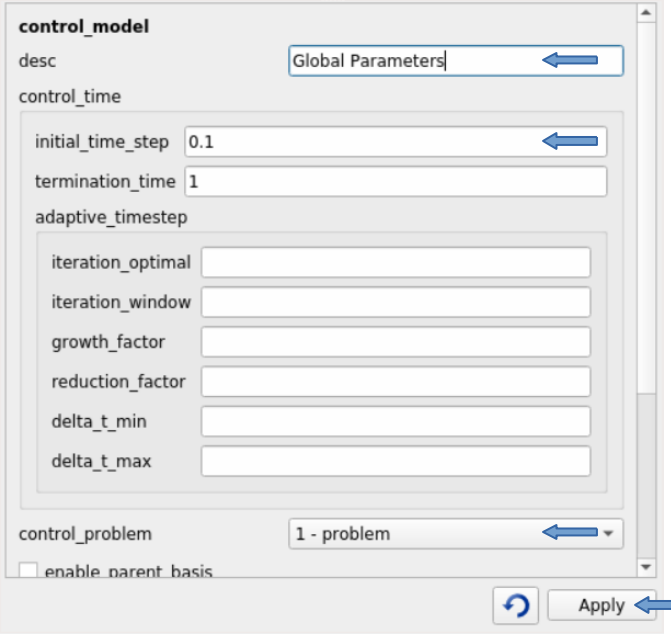
Add timestep cards



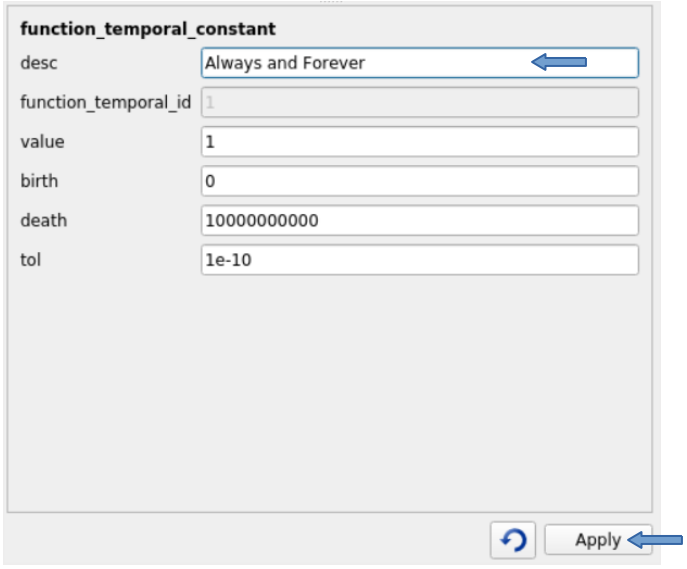


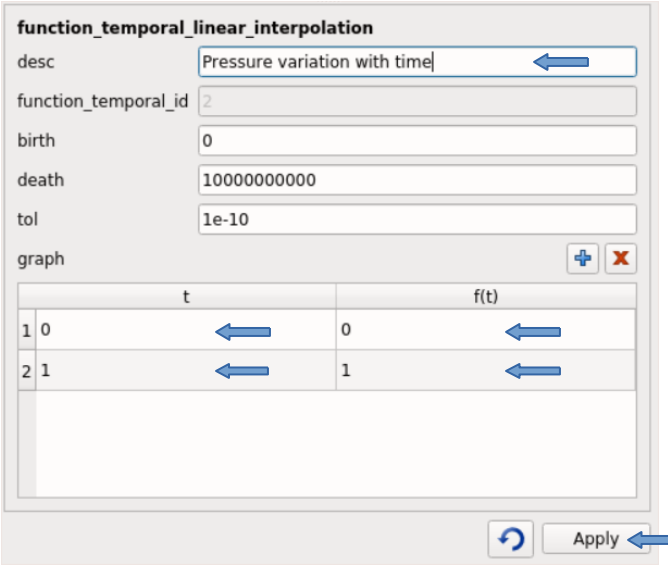
Add problem and control_model cards



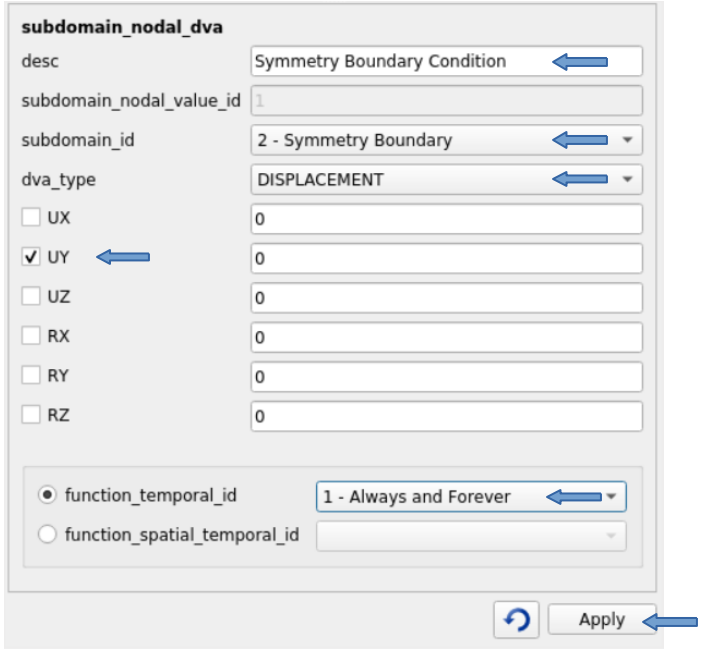


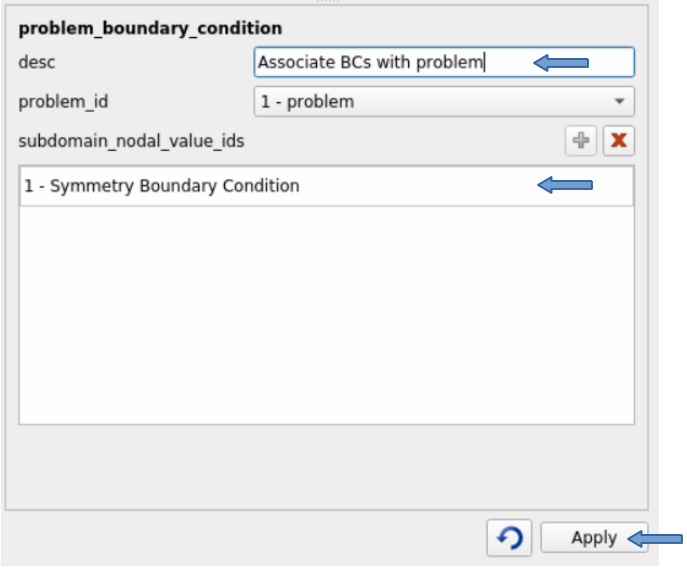
Add temporal functions



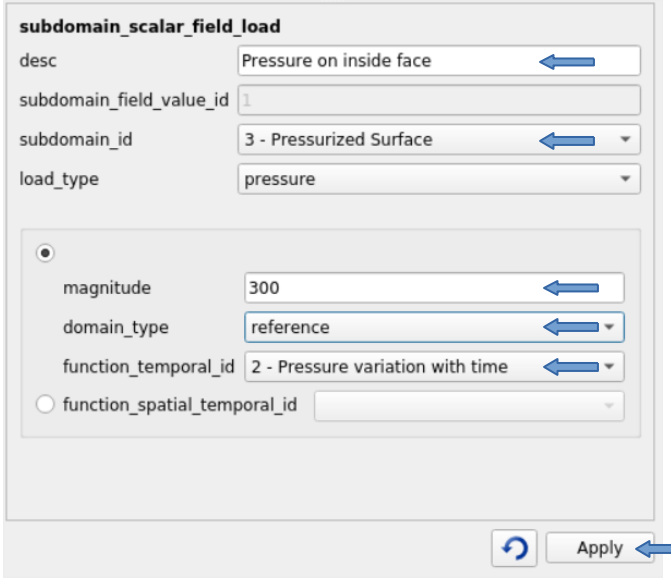


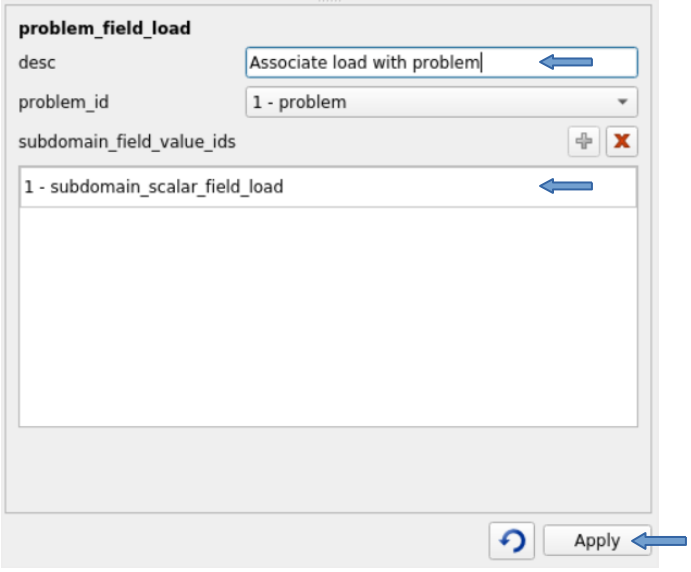
Add symmetry boundary condition



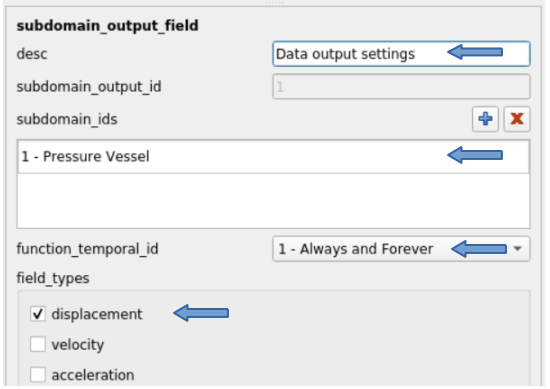


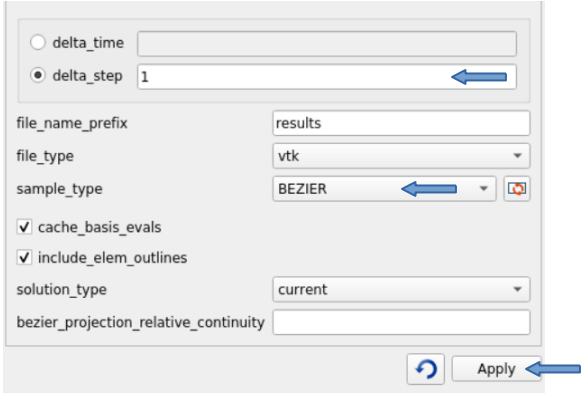
Add pressure load





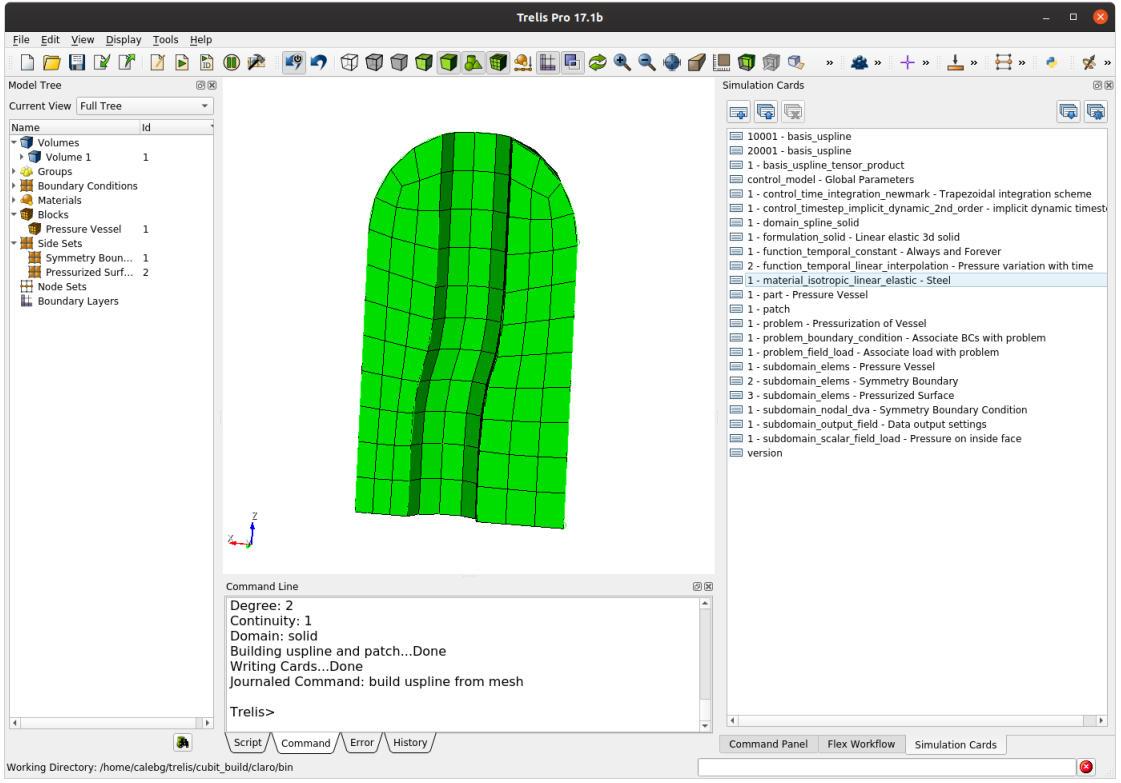
Add output


subdomain_output_field
desc
Data output settings
subdomain_output_id
1
subdomain_ids
1 - Pressure Vessel
function_temporal_id
1 - Always and Forever
field_types
displacement
velocity
acceleration

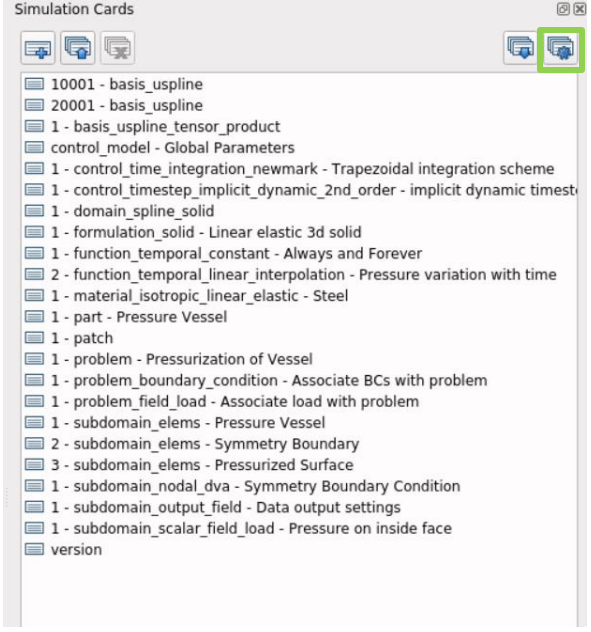

delta_time
delta_step
1
file_name_prefix
results
file_type
vtk
sample_type
BEZIER
cache_basis_evals
include_elem_outlines
solution_type
current
bezier_projection_relative_continuity
Apply

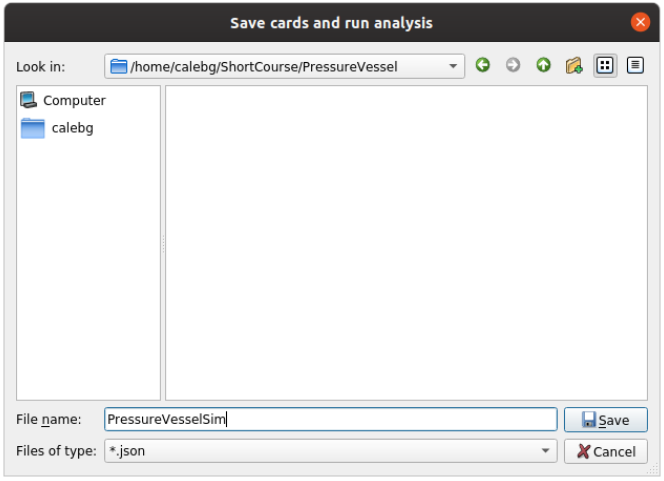
Input finished

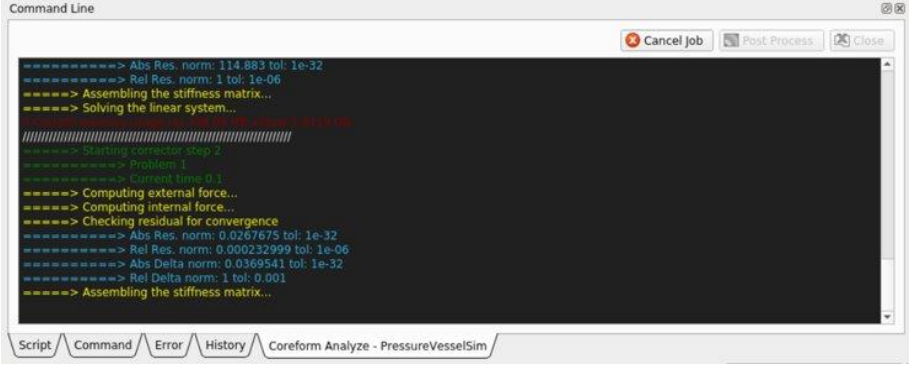
Your window should now look like this:



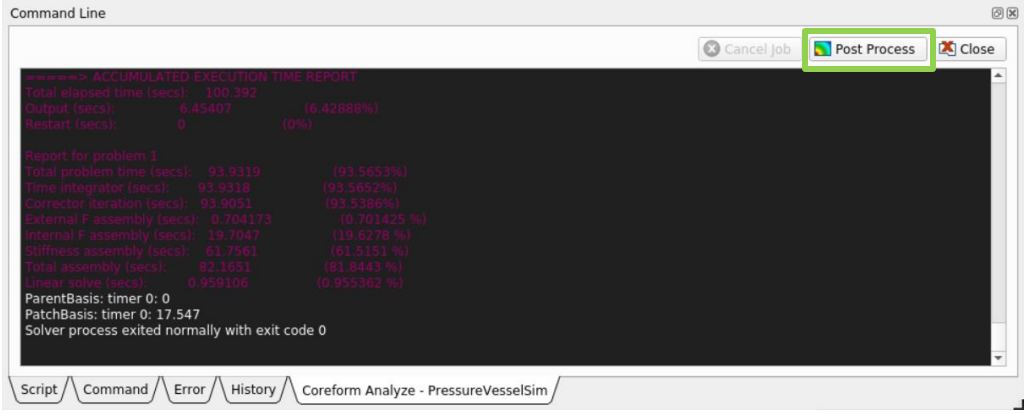
Run Simulation



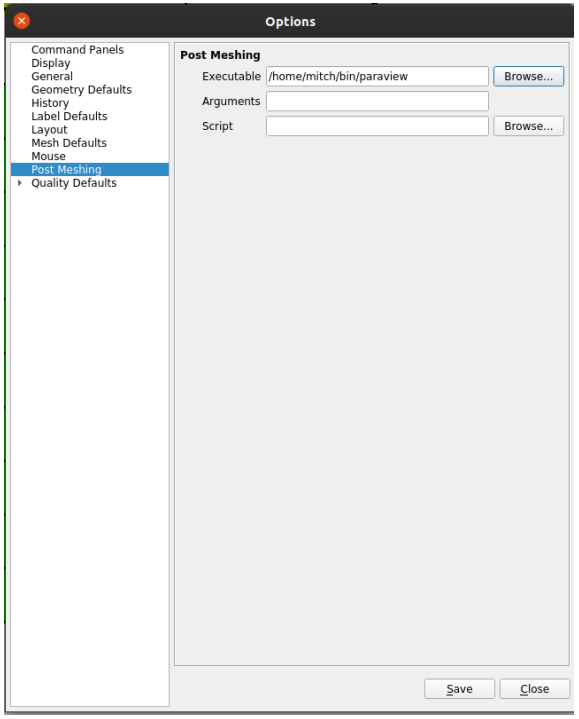




View Results

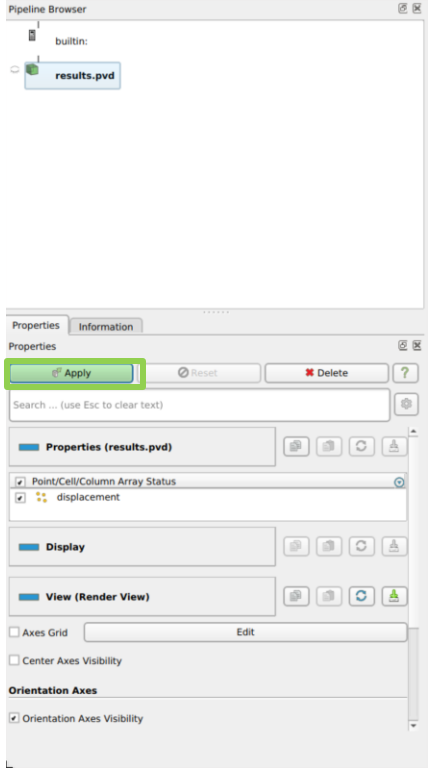


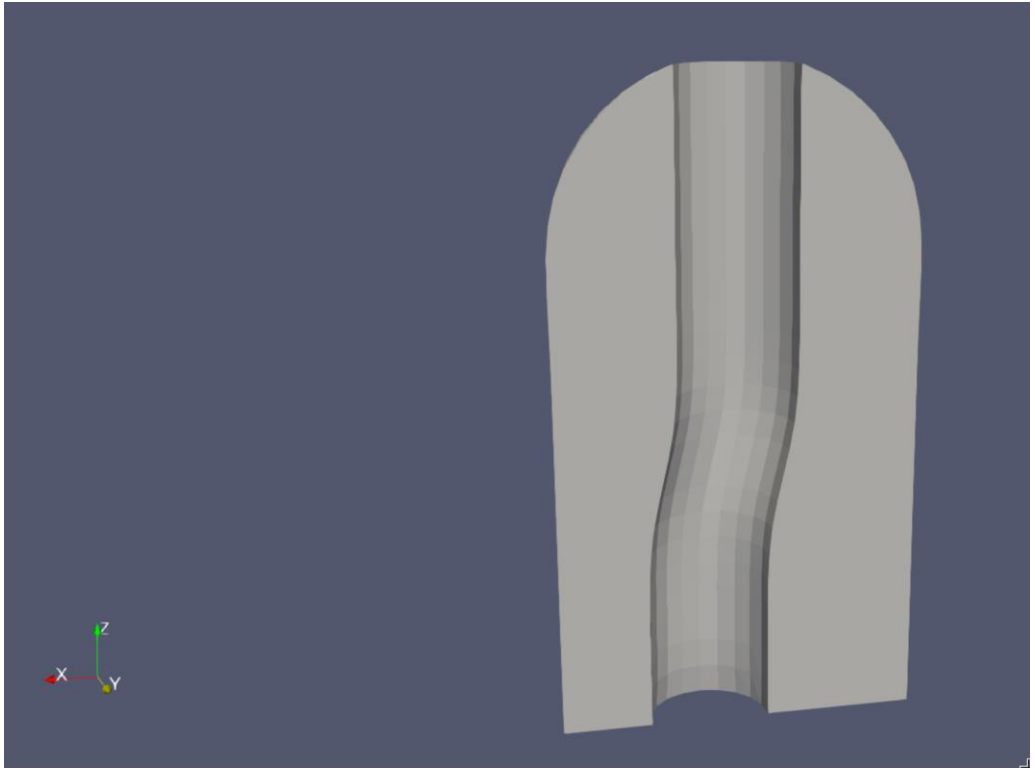
Set Post Process Executable



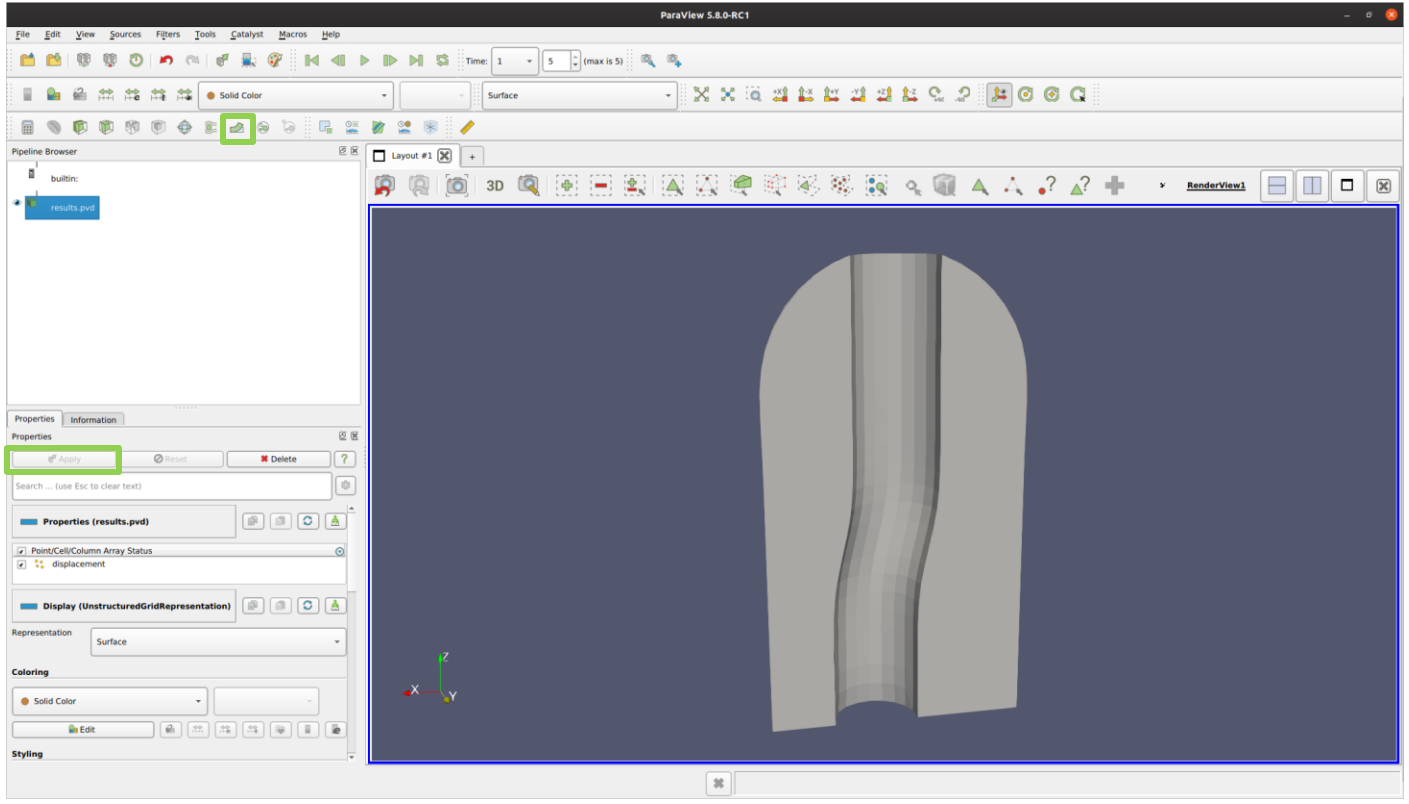
Tools -> Options -> Post Meshing

View Results

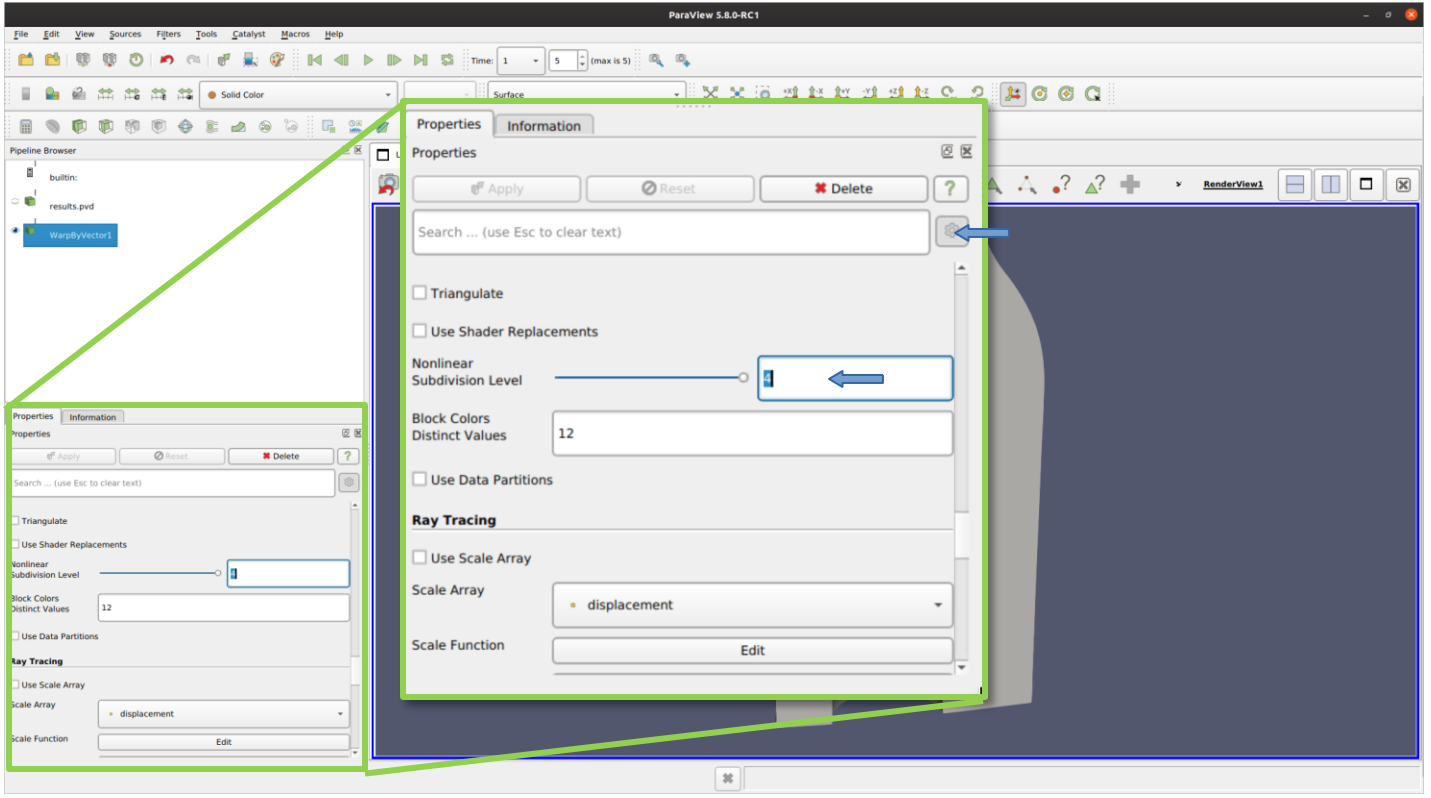




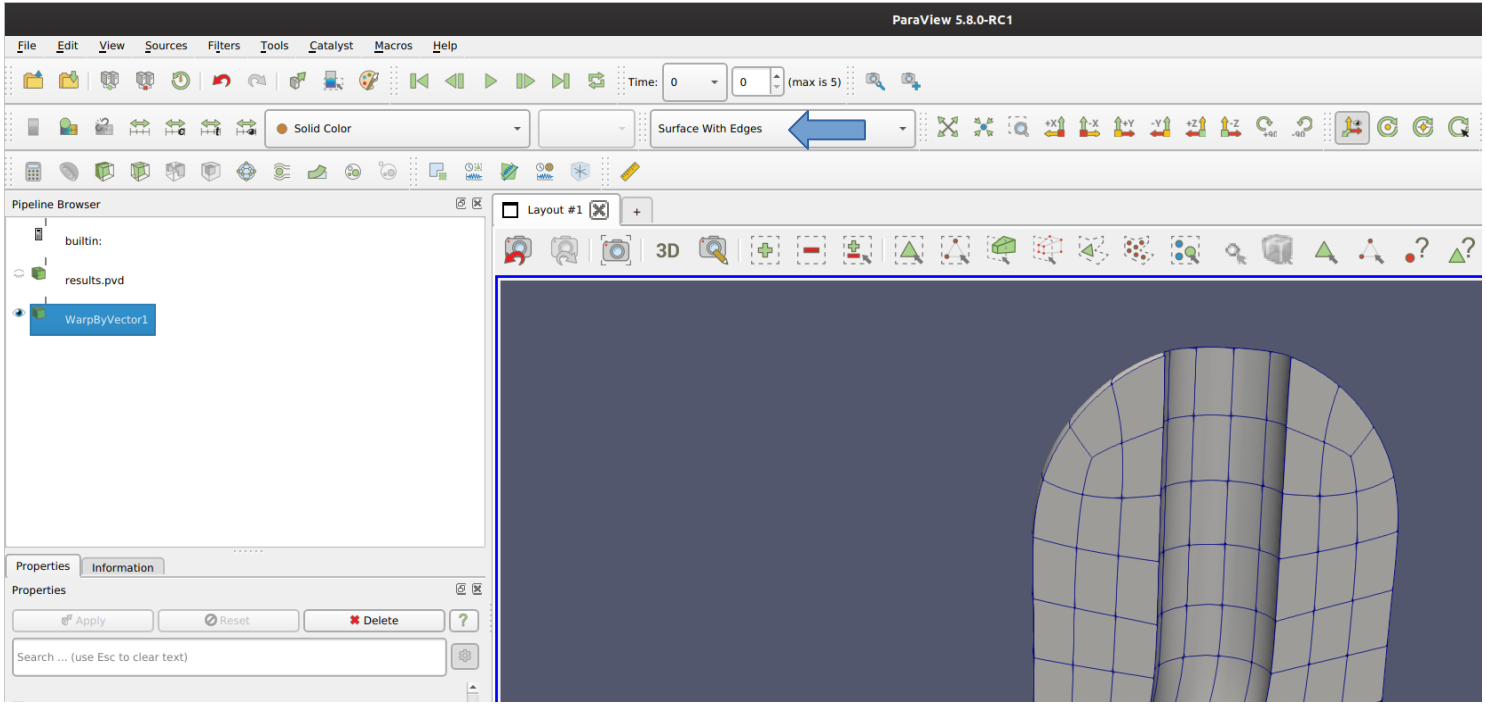
Warp By Vector



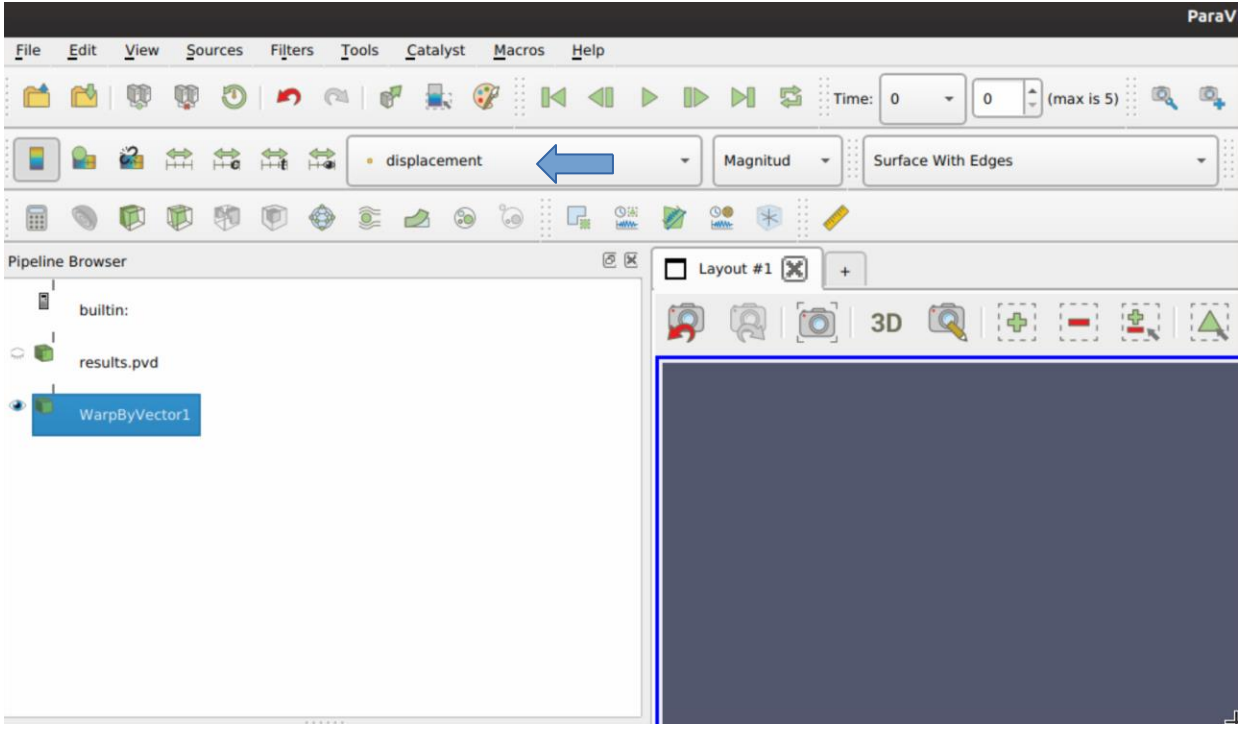
View Smooth Spline



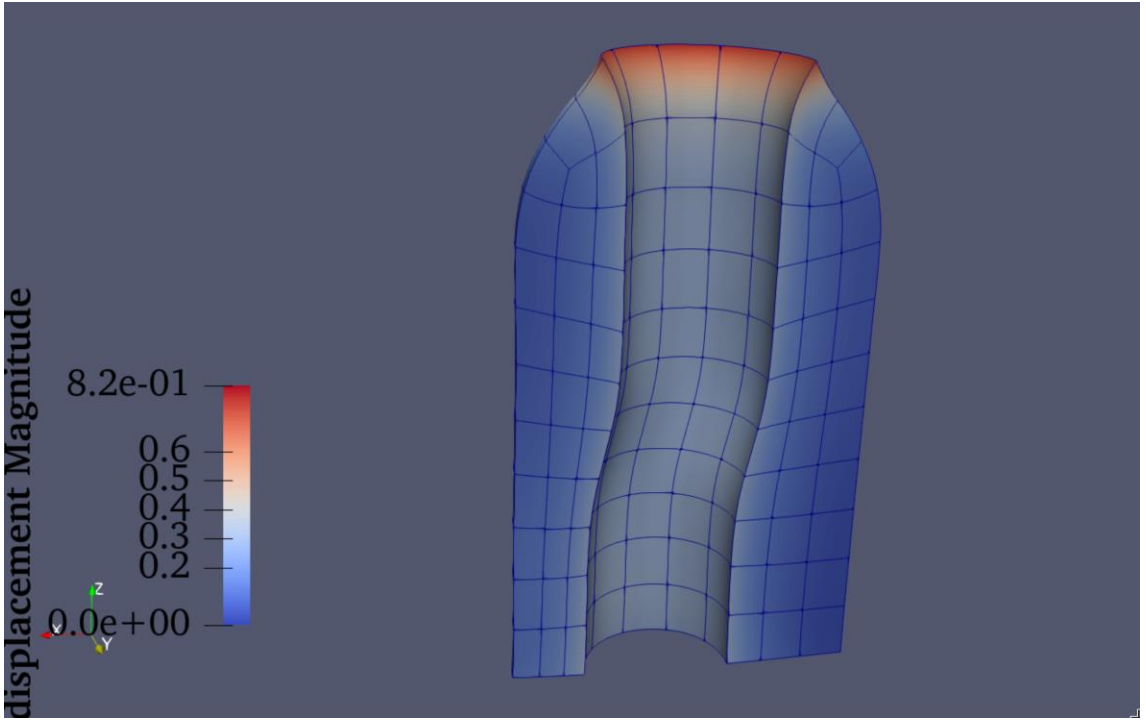
View Mesh


ParaView 5.8.0-RC1
File
Edit
View
Sources
Filters
Tools
Catalyst
Macros
Help
Time:
0
(max is 5)
Solid Color
Surface With Edges
Pipeline Browser
builtin:
results.pvd
WarpByVector1
Layout #1
3D
Properties
Information
Properties
Apply
Reset
Delete
Search ... (use Esc to clear text)

View Displacements

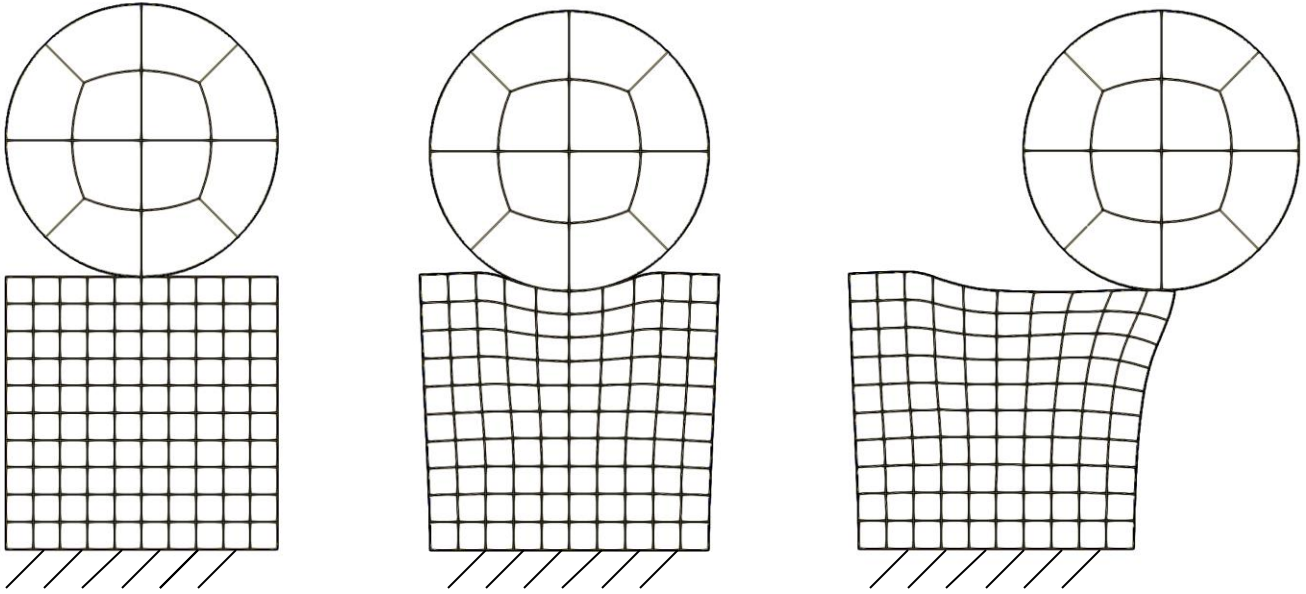


Play Displacements



You may need to adjust the color scaling

Problem 2: Deep Rolling



This tutorial will simulate contact between a disc and a block as shown on the left. The material model used for both parts will allow for plastic deformation

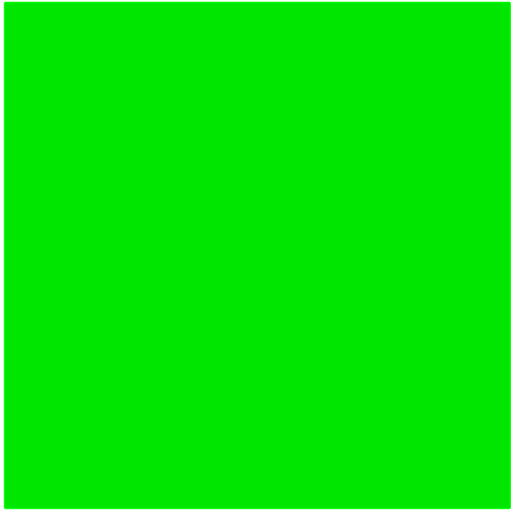
The problem will be modeled as a 2D solid quasi-static simulation

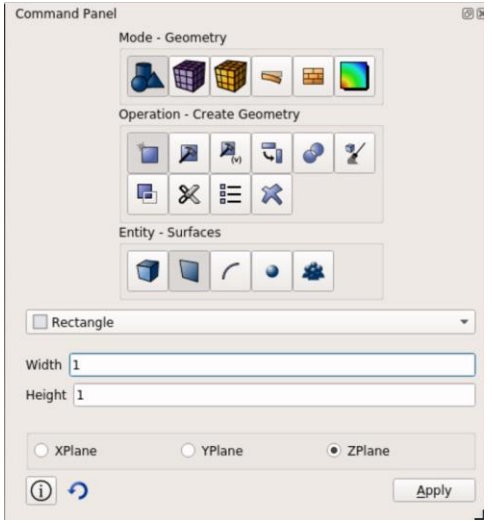
Learning Objectives

Tutorial participants will learn:

- How to manage a simulation with multiple parts
- How to set up displacement boundary conditions
- How to use material models with plasticity
- How to set up contact

Create block

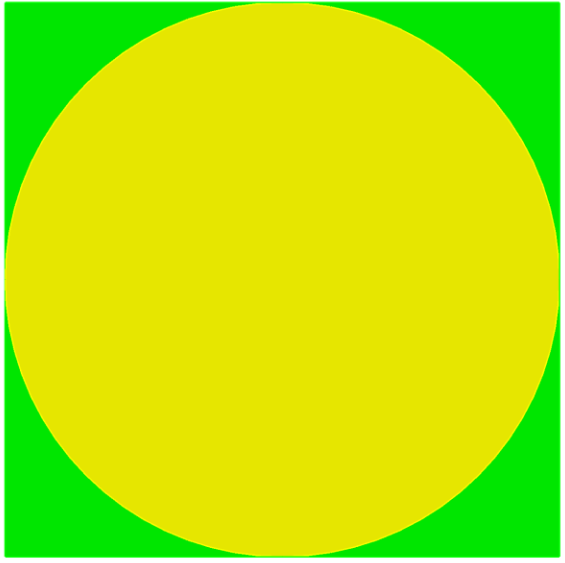


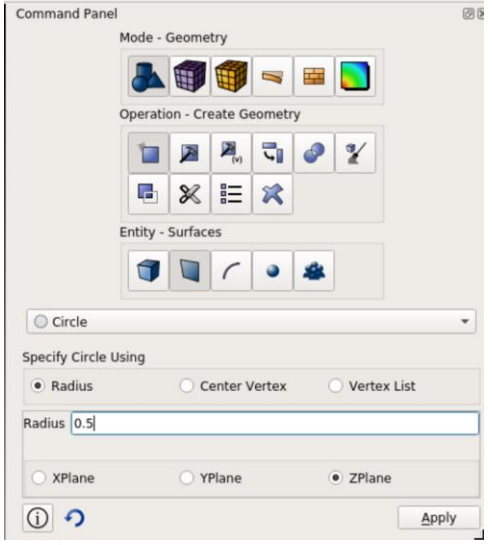


- or -

Create surface rectangle width 1 height 1 zplane

Create disc

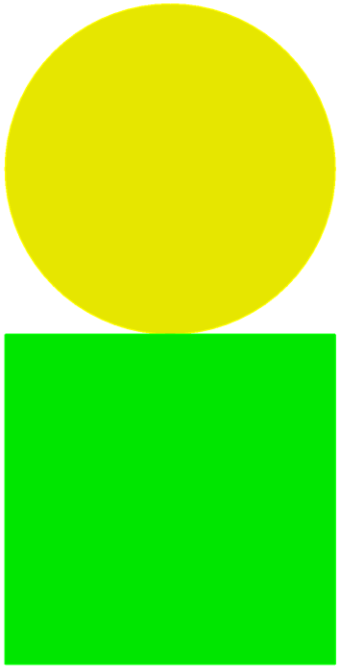


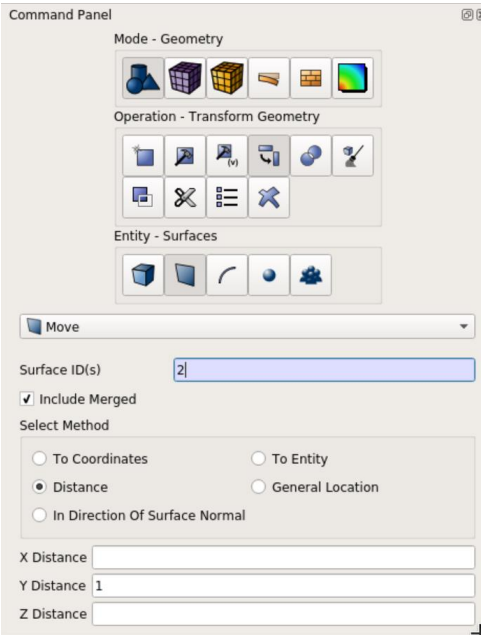


- or -

Create surface circle radius 0.5 zplane

Move disc



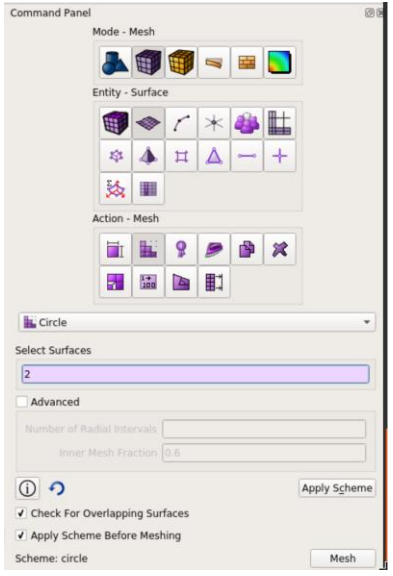


- or -

This geometry is available at this address:
https://coreform.com/shortcourse/deep_rolling_init.cub

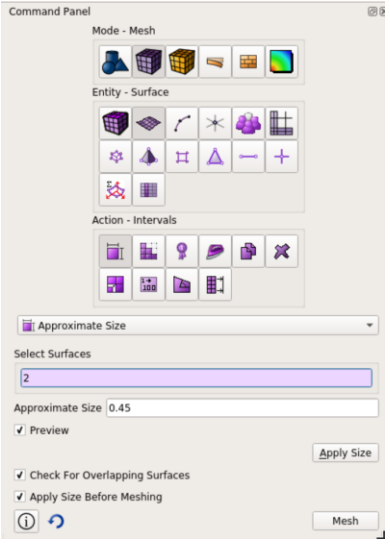
Move surface 2 y 1

Mesh disc



- or -

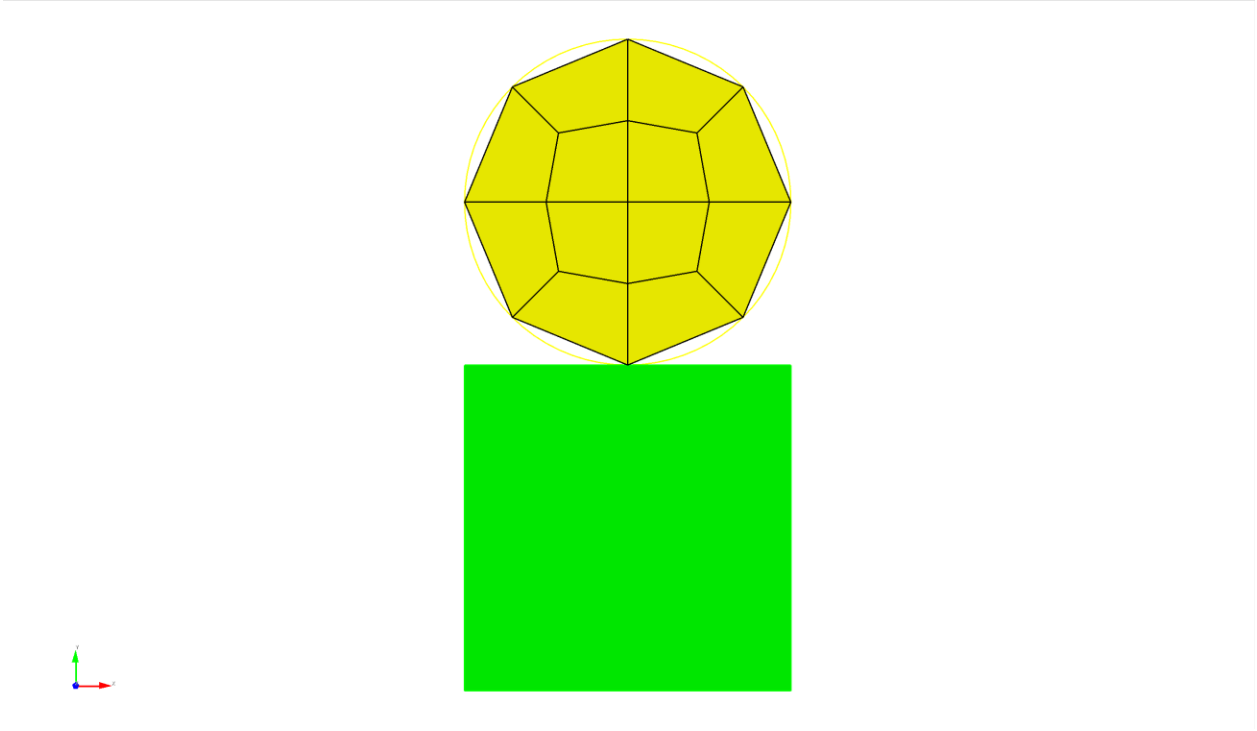
Surface 2 scheme circle



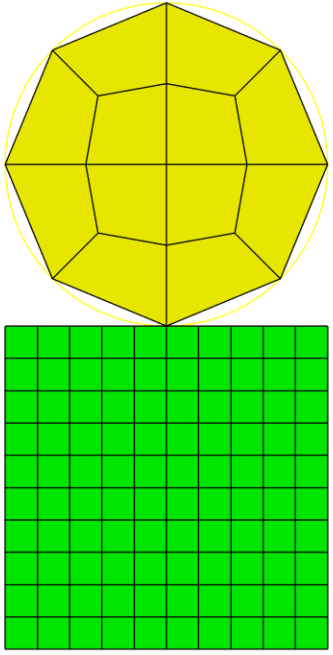
- or -

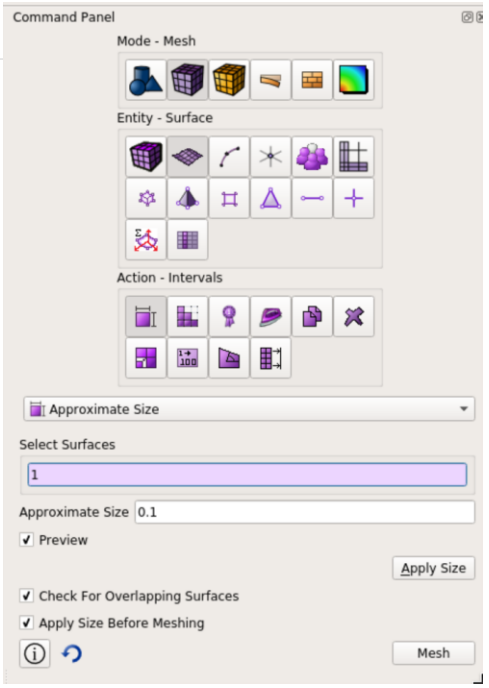
Surface 2 size 0.45
Mesh surface 2

Mesh disc



Mesh block

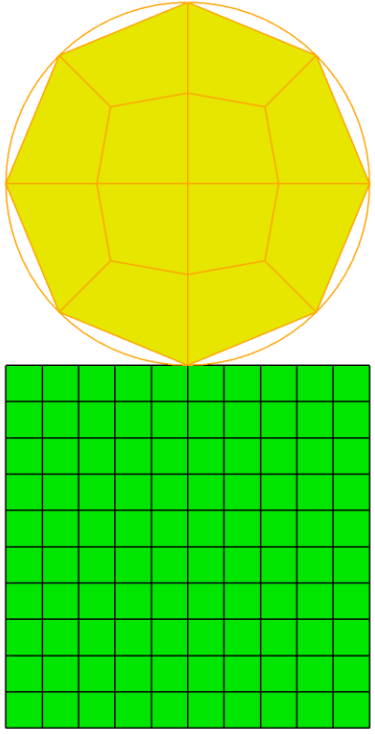


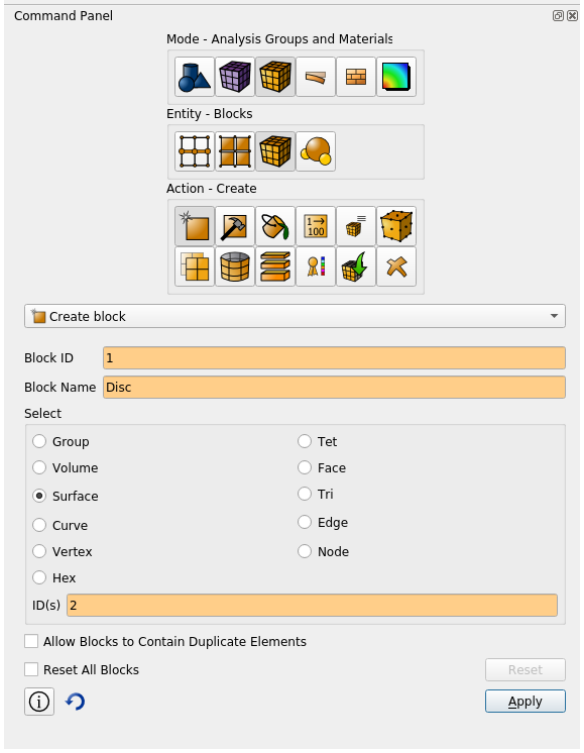


- or -

Surface 1 size 0.1
Mesh surface 1

Create block sets - Disc



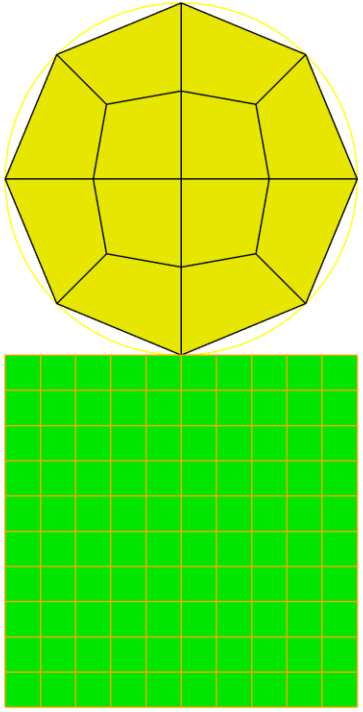


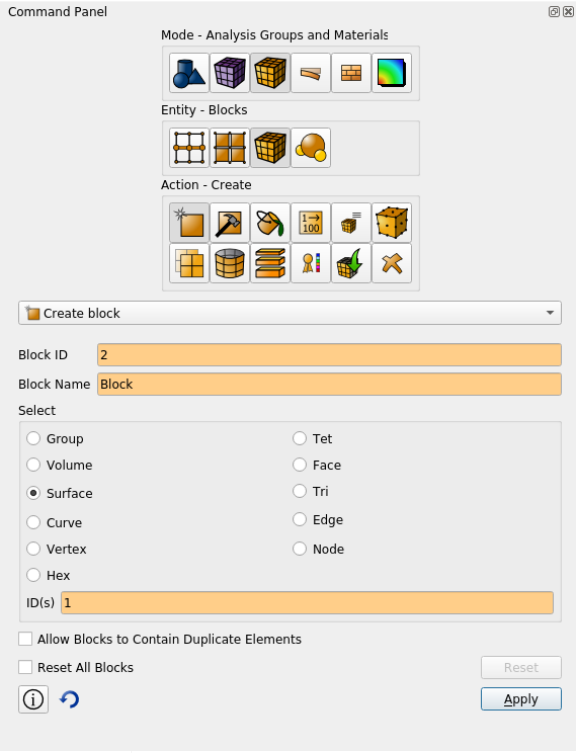
- or -

Block 1 add surface 2

Block 1 name "Disc"

Create block sets - Block

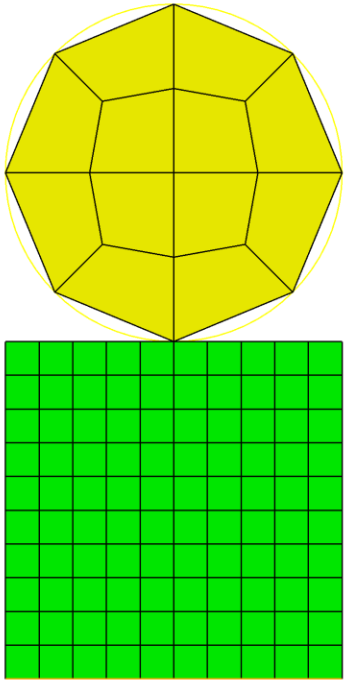


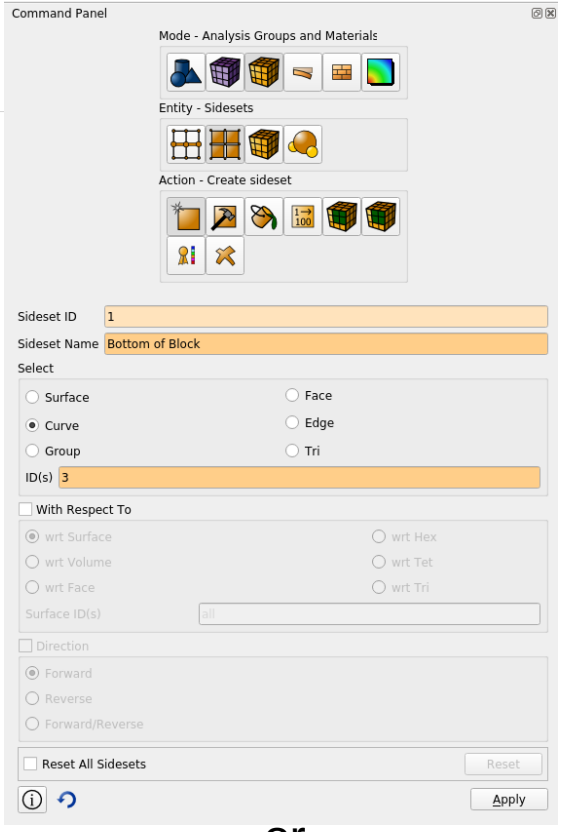


- or -

Block 2 add surface 1
Block 2 name "Block"

Create side sets bottom of block

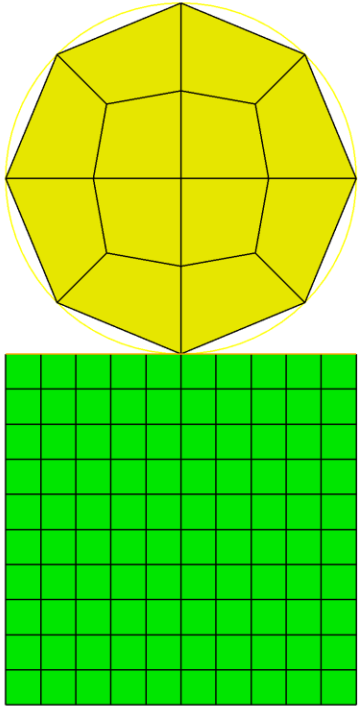


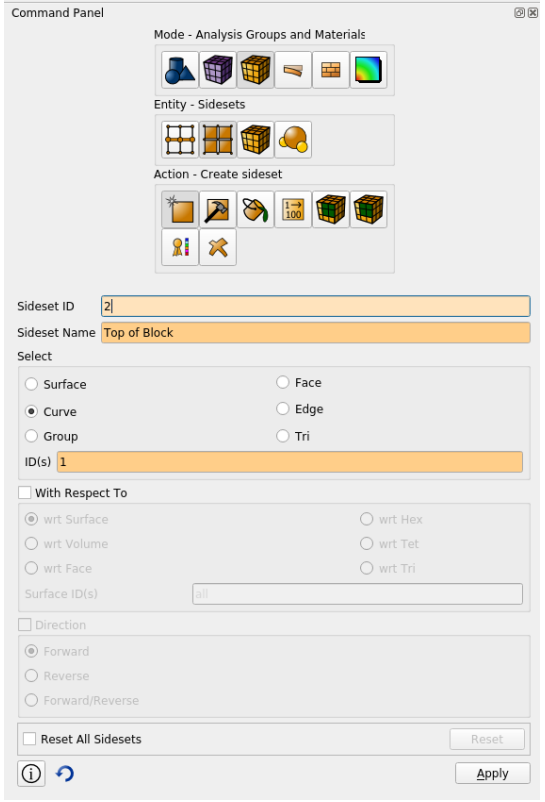


- or -

Sideset 1 add curve 3
Sideset 1 name "Bottom of Block"

Create side sets top of block

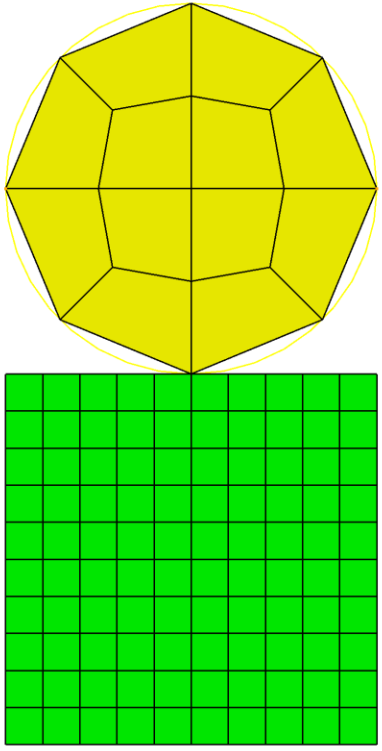




- or -

Sideset 2 add curve 1
Sideset 2 name "Top of Block"

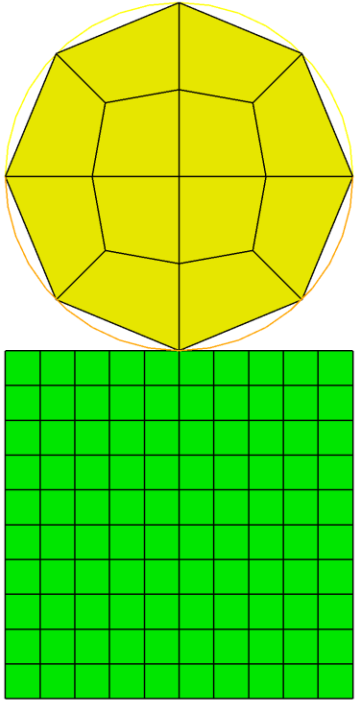
Split disc curve

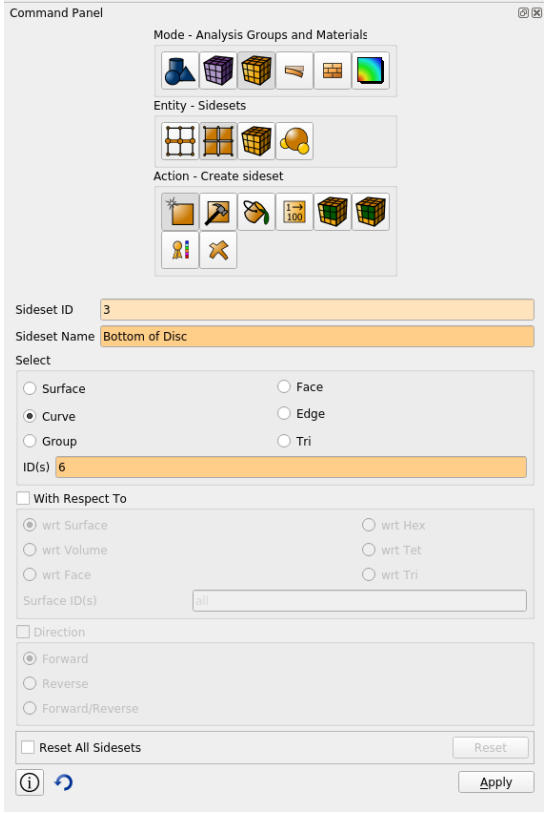


- or -

Partition create curve 5 node 1 5

Create side sets bottom of disc

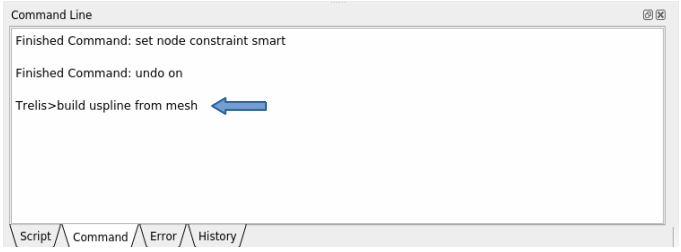




- or -

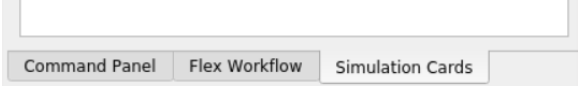
Sideset 3 add curve 6
Sideset 3 name "Bottom of Disc"

Generate and view simulation geometry cards

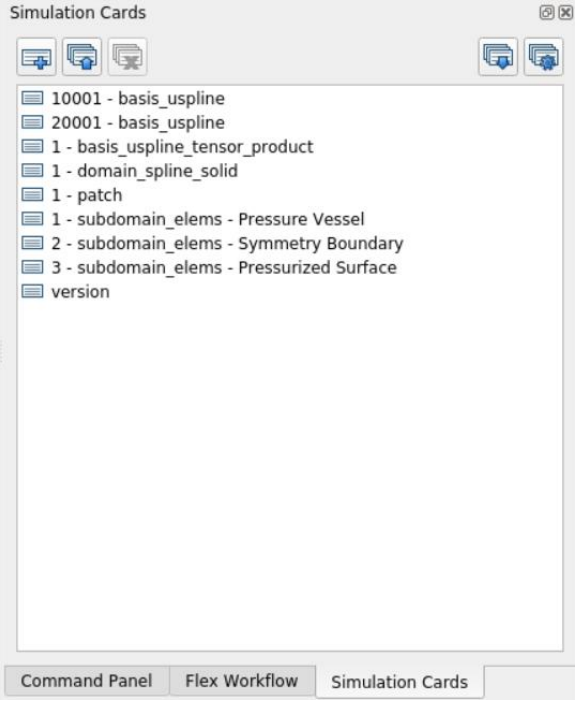


Build is finished when Trelis> prompt returns.

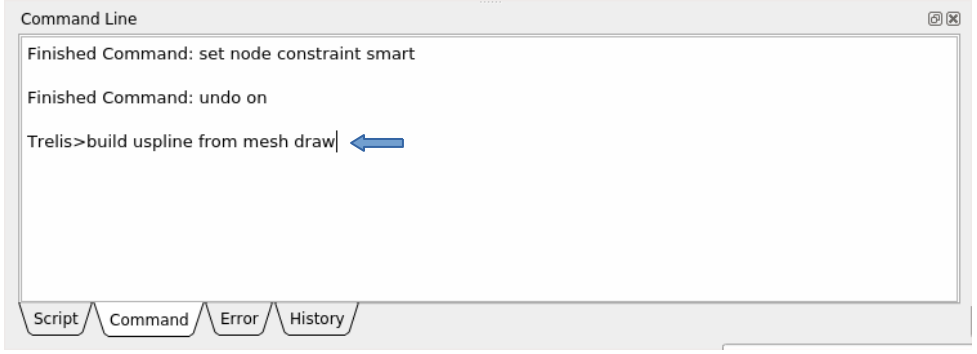
Select "Simulation Cards" tab in bottom left



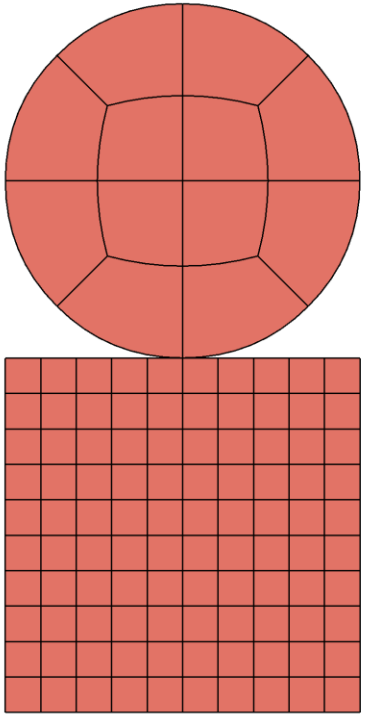
The simulation cards tab should now look like this:



Build U-spline

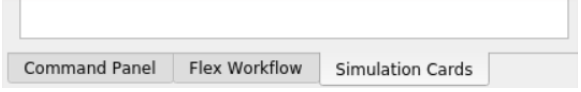


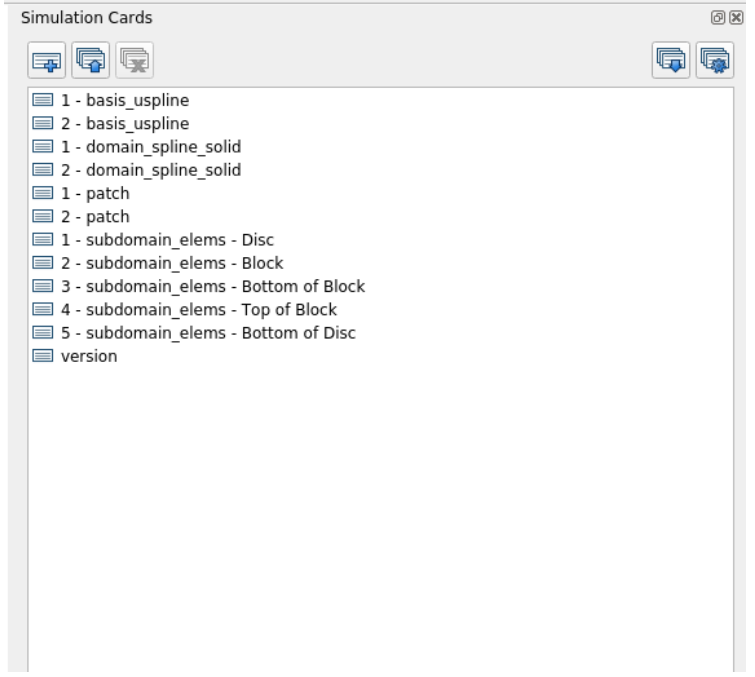
Build is finished when Trelis> prompt returns.



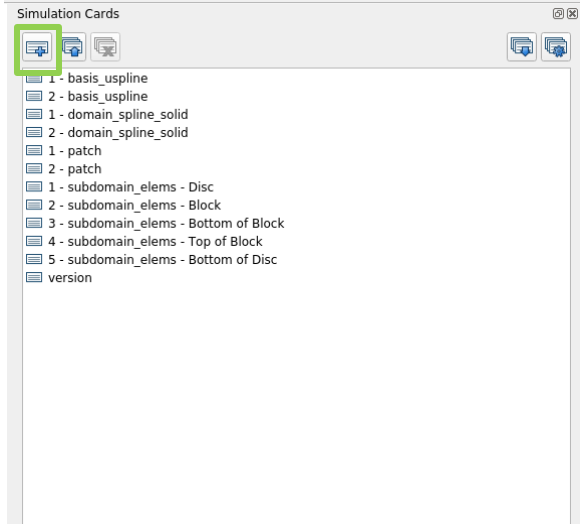
View simulation cards

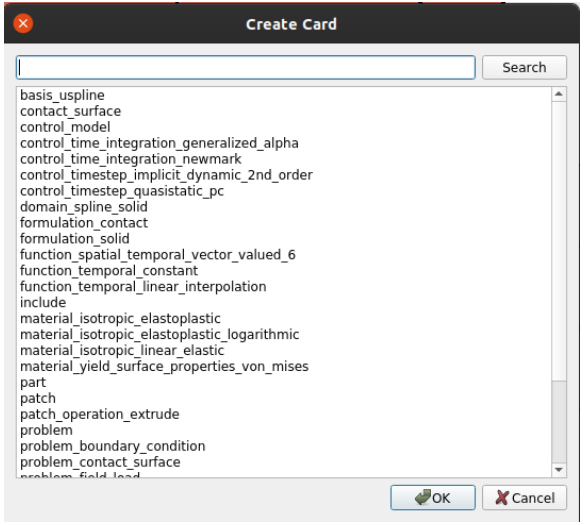
Select "Simulation Cards" tab in bottom right



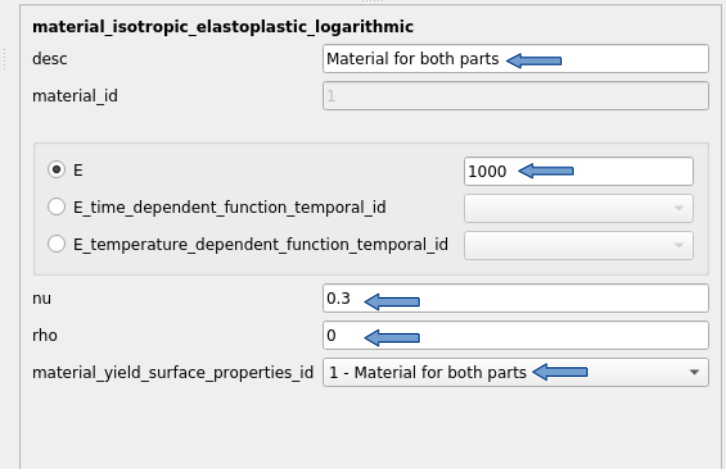


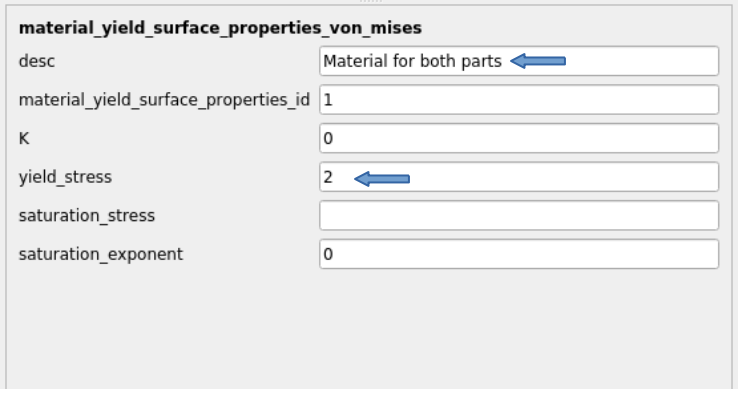
Creating Cards



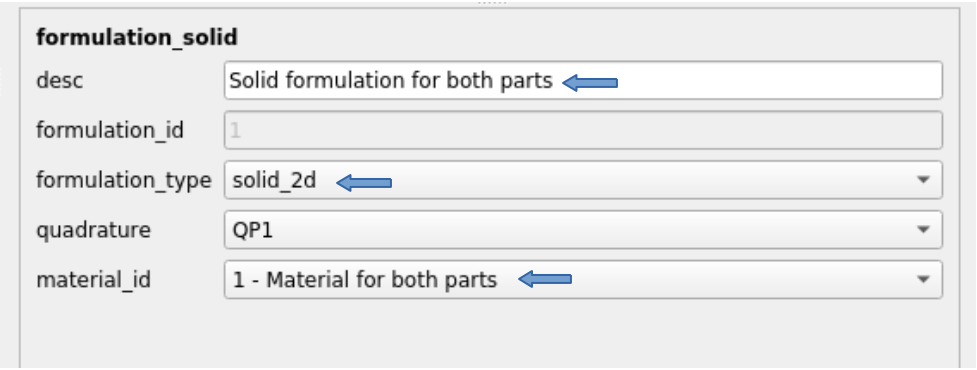


Material definition

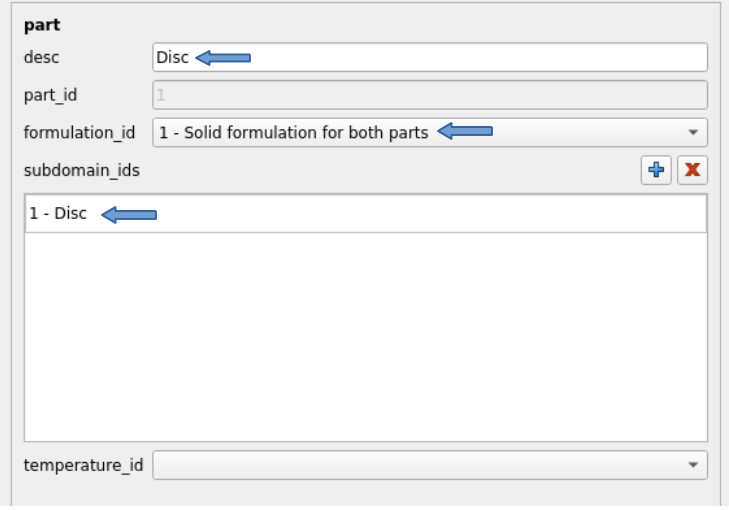


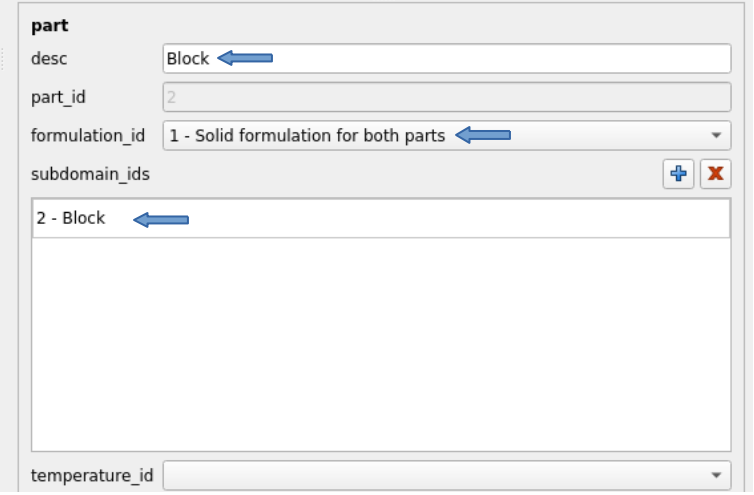


Element Formulation

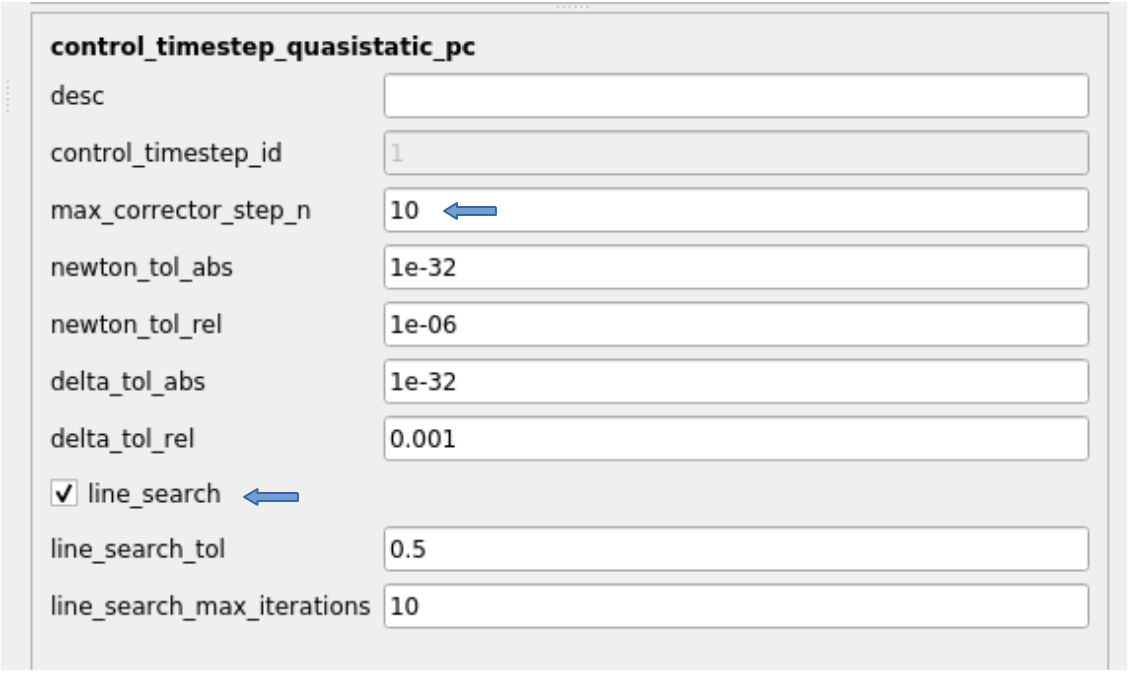


Parts

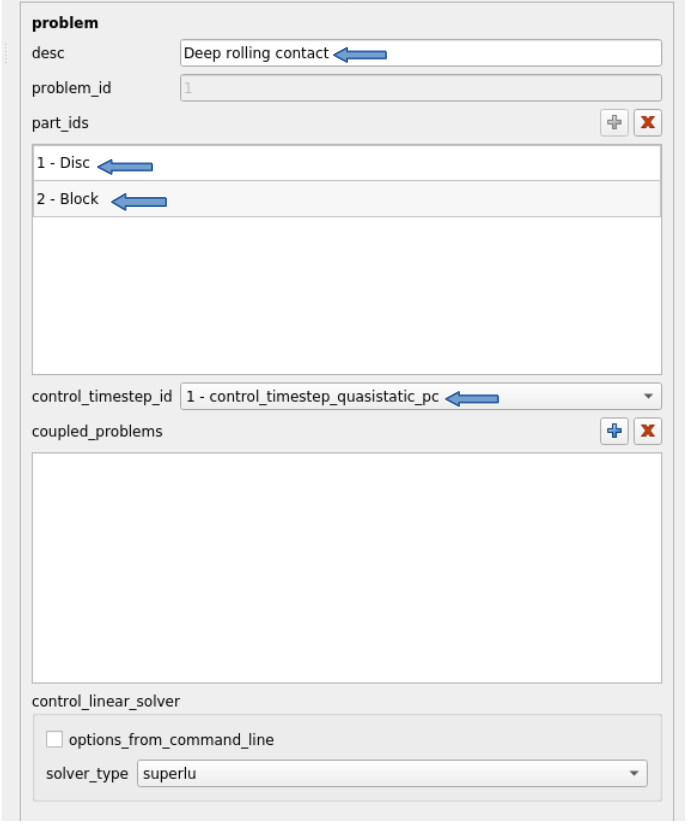

part
desc
Disc
part_id
1
formulation_id
1 - Solid formulation for both parts
subdomain_ids
1 - Disc
temperature_id

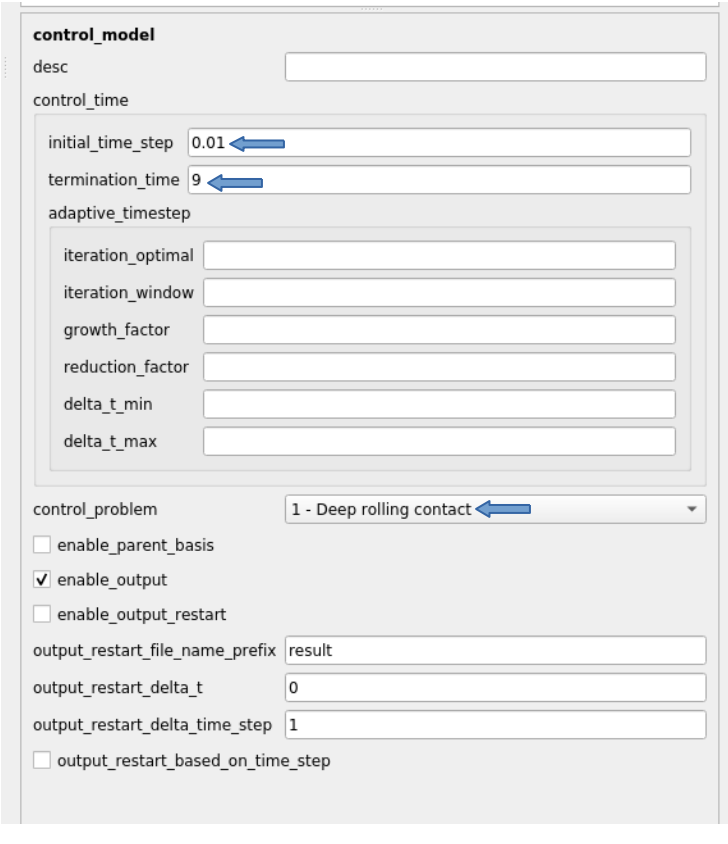

part
desc
Block
part_id
2
formulation_id
1 - Solid formulation for both parts
subdomain_ids
2 - Block
temperature_id

Control Timestep



Problem and Control Model





Downward displacement of disc

function_temporal_linear_interpolation

desc	Downward displacement of disc
function_temporal_id	1
birth	0
death	10000000000
tol	1e-10

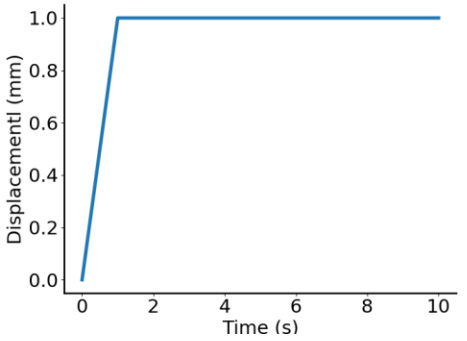
graph

	t	f(t)
1	0	0
2	1	1
3	10	1

subdomain_nodal_dva

desc	Downward displacement of disc
subdomain_nodal_value_id	1
subdomain_id	1 - Disc
dva_type	DISPLACEMENT
UX	0
UY	-0.075
UZ	0
RX	0
RY	0
RZ	0

- function_temporal_id: 1 - Downward displacement of disc
- function_spatial_temporal_id:



Right displacement of disc

function_temporal_linear_interpolation

desc	Right displacement of disc
function_temporal_id	2
birth	0
death	10000000000
tol	1e-10

graph

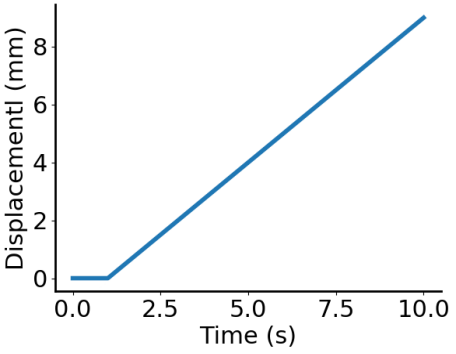
	t	f(t)
1	0	0
2	1	0
3	2	1
4	10	9

subdomain_nodal_dva

desc	Right displacment of Disc
subdomain_nodal_value_id	2
subdomain_id	1 - Disc
dva_type	DISPLACEMENT
UX	0.075
UY	0
UZ	0
RX	0
RY	0
RZ	0

function_temporal_id: 2 - Right displacement of disc

function_spatial_temporal_id:



Clamp bottom of block

function_temporal_constant

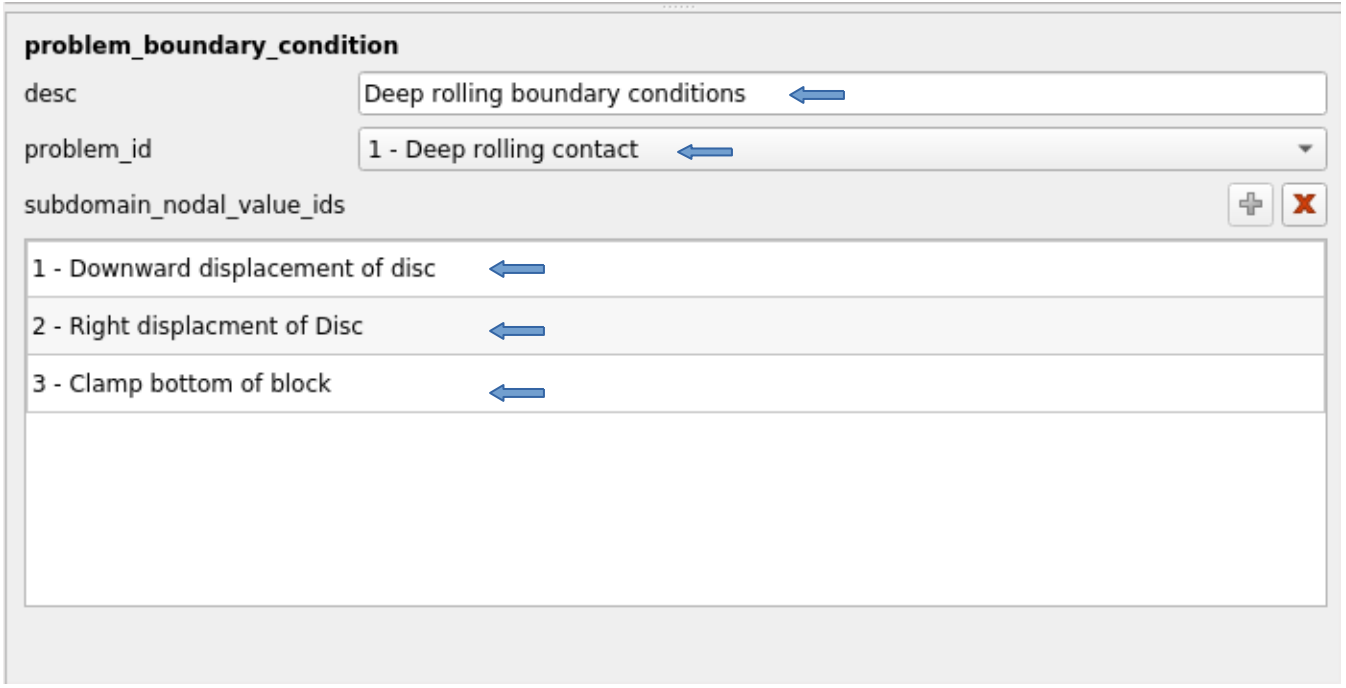
desc	Always 1
function_temporal_id	3
value	1
birth	0
death	10000000000
tol	1e-10

subdomain_nodal_dva

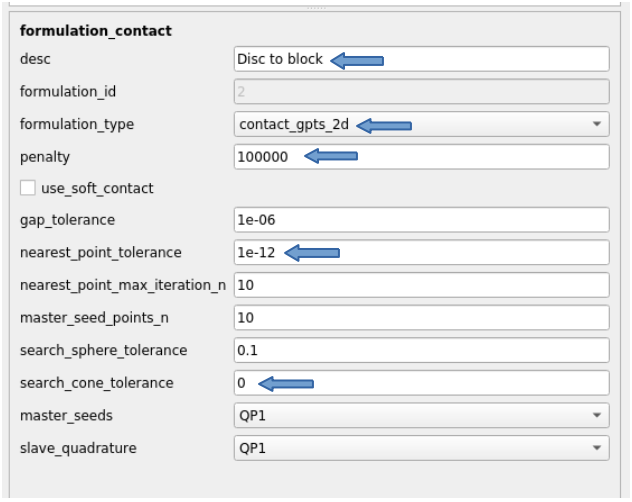
desc	Clamp bottom of block
subdomain_nodal_value_id	3
subdomain_id	3 - Bottom of Block
dva_type	DISPLACEMENT
☑ UX	0
☑ UY	0
☐ UZ	0
☐ RX	0
☐ RY	0
☐ RZ	0

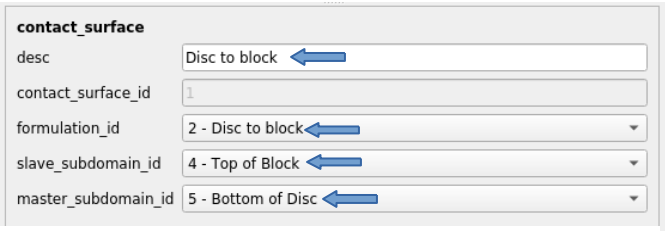
◉ function_temporal_id	3 - Always 1
○ function_spatial_temporal_id	

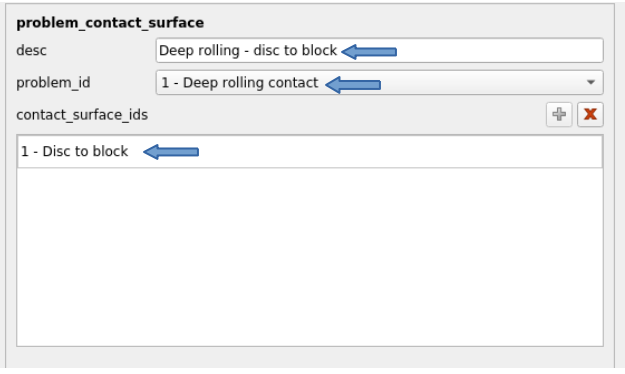
Problem boundary condition



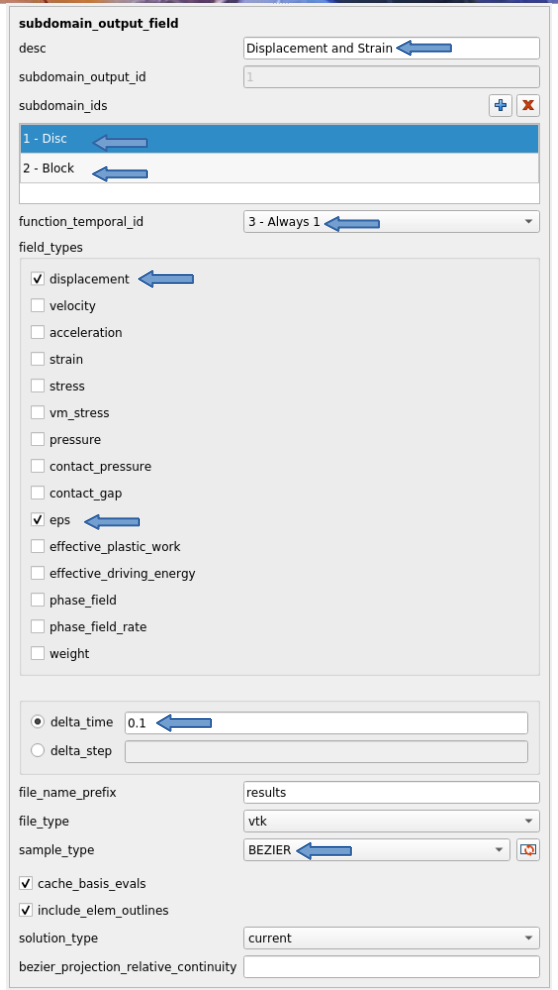
Set up contact



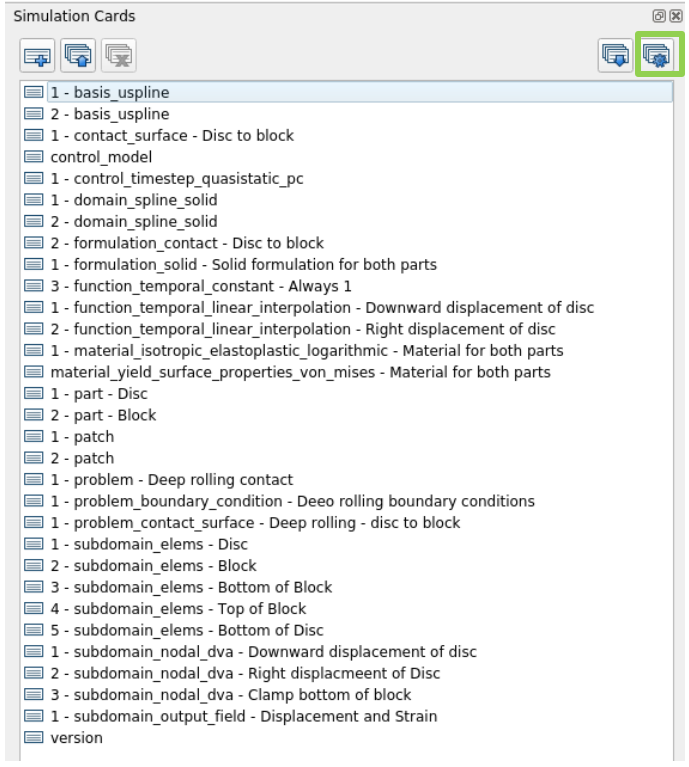


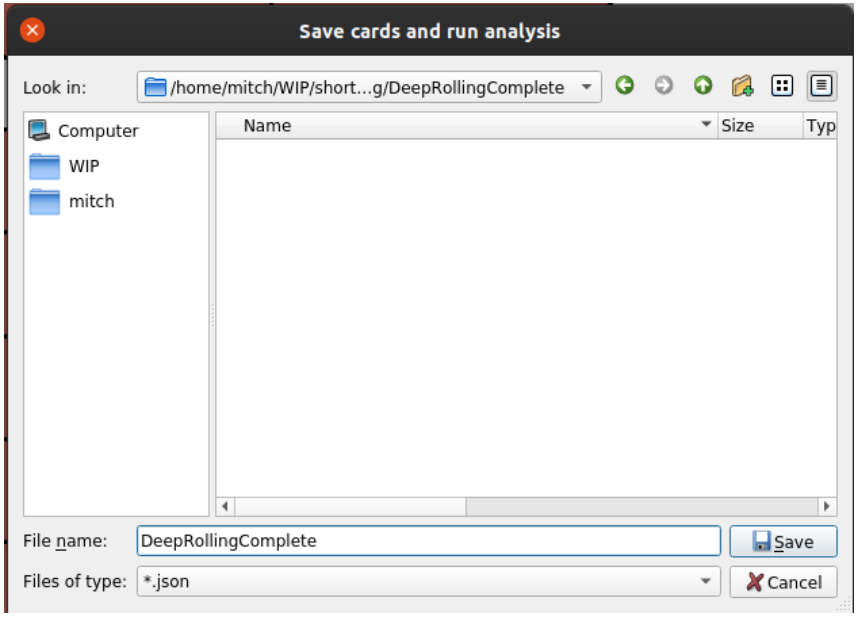


Output


subdomain_output_field
desc
Displacement and Strain
subdomain_output_id
1
subdomain_ids
1 - Disc
2 - Block
function_temporal_id
3 - Always 1
field_types
displacement
velocity
acceleration
strain
stress
vm_stress
pressure
contact_pressure
contact_gap
eps
effective_plastic_work
effective_driving_energy
phase_field
phase_field_rate
weight
delta_time
0.1
delta_step
file_name_prefix
results
file_type
vtk
sample_type
BEZIER
cache_basis_evals
include_elem_outlines
solution_type
current
bezier_projection_relative_continuity

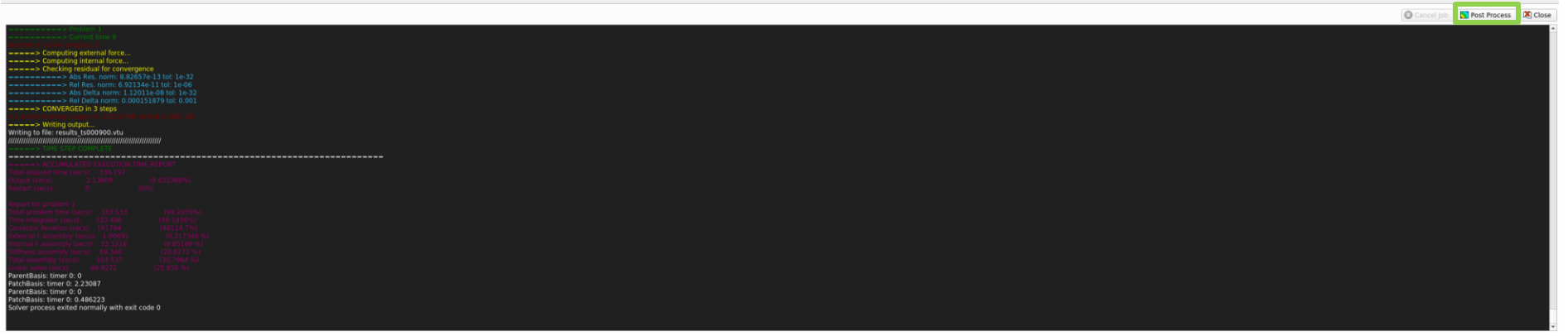
Run Simulation



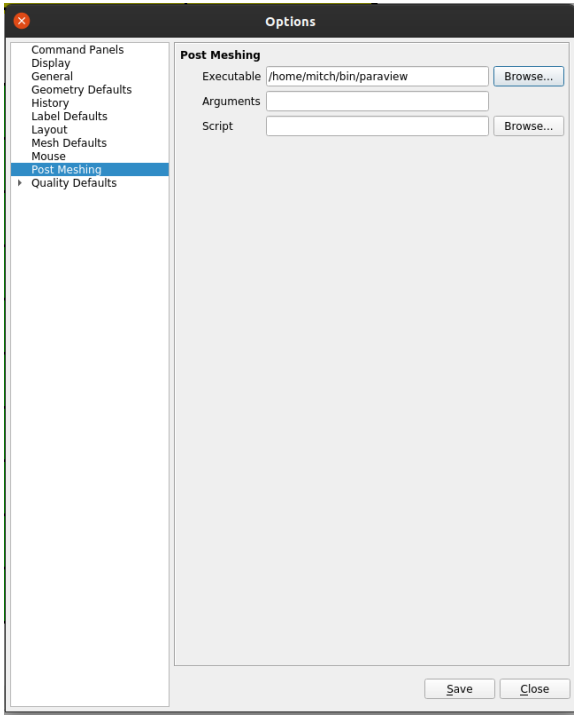


Run Simulation

View Results

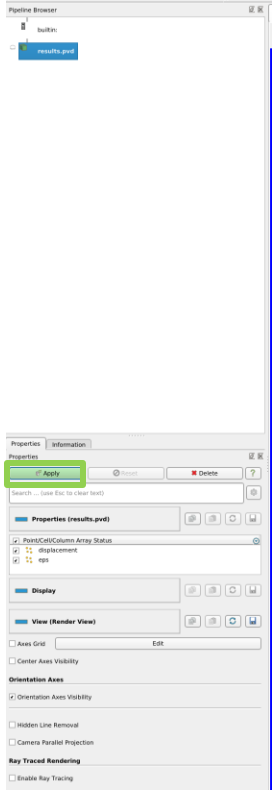


Set Post Process Executable

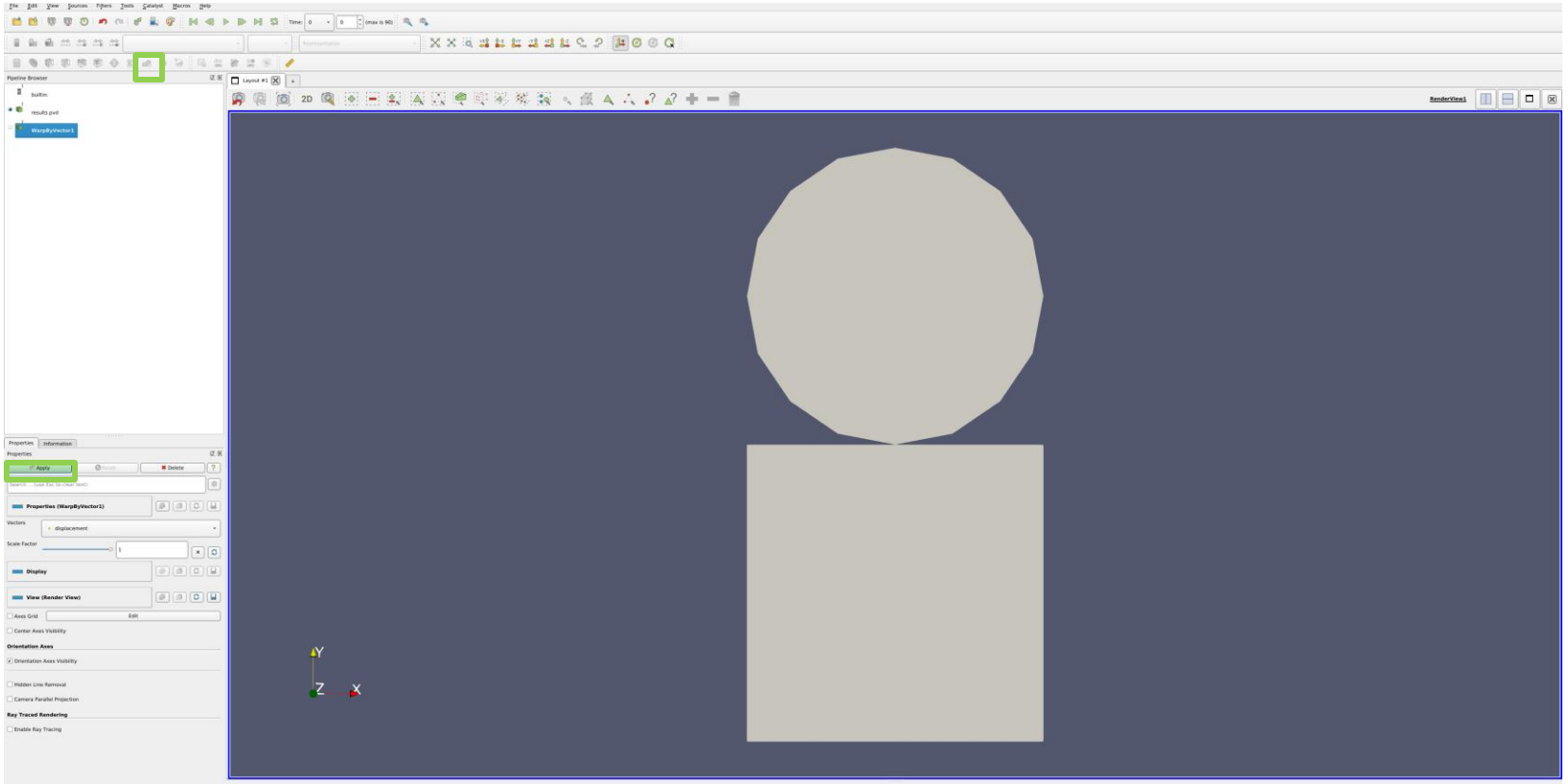


Tools -> Options -> Post Meshing

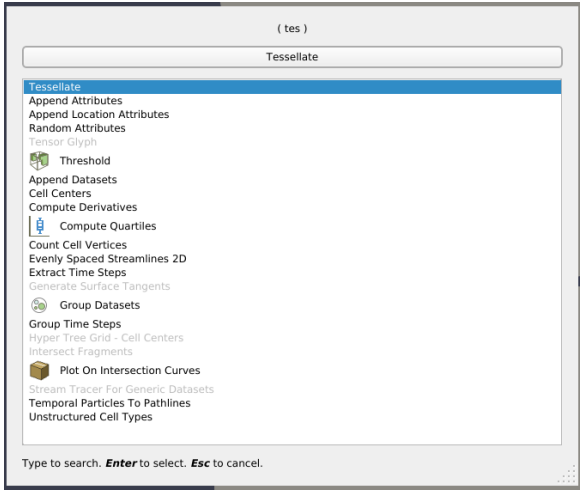
View Results



Warp By Vector

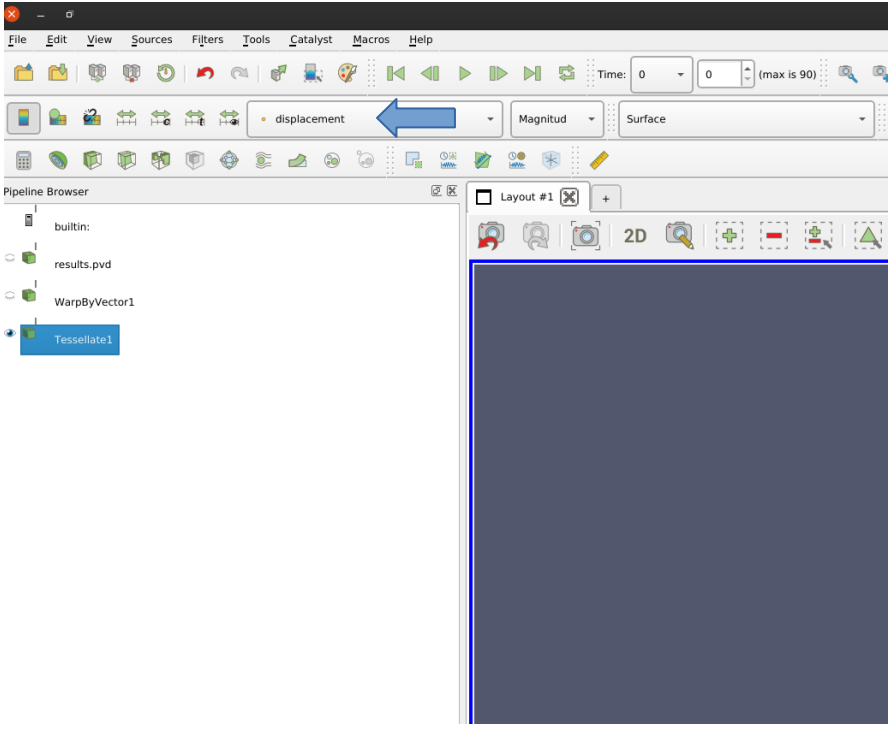


Tessellate

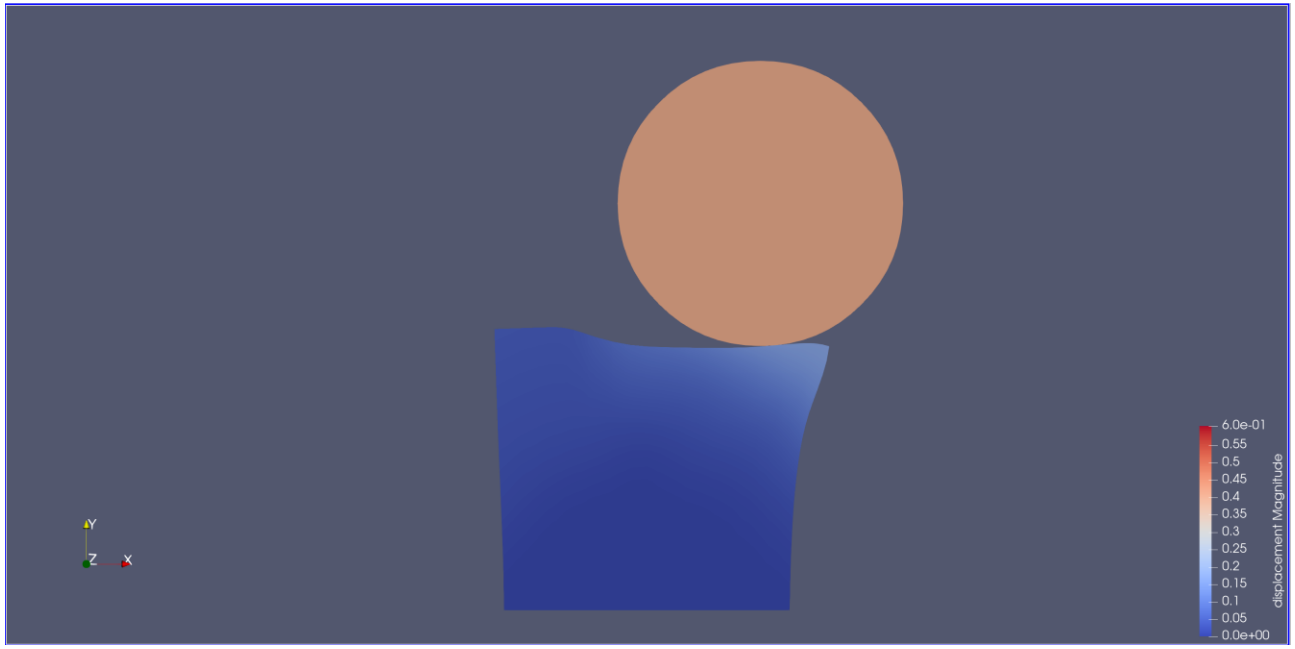


Filters -> Search -> Tessellate
Then click apply

View Displacements

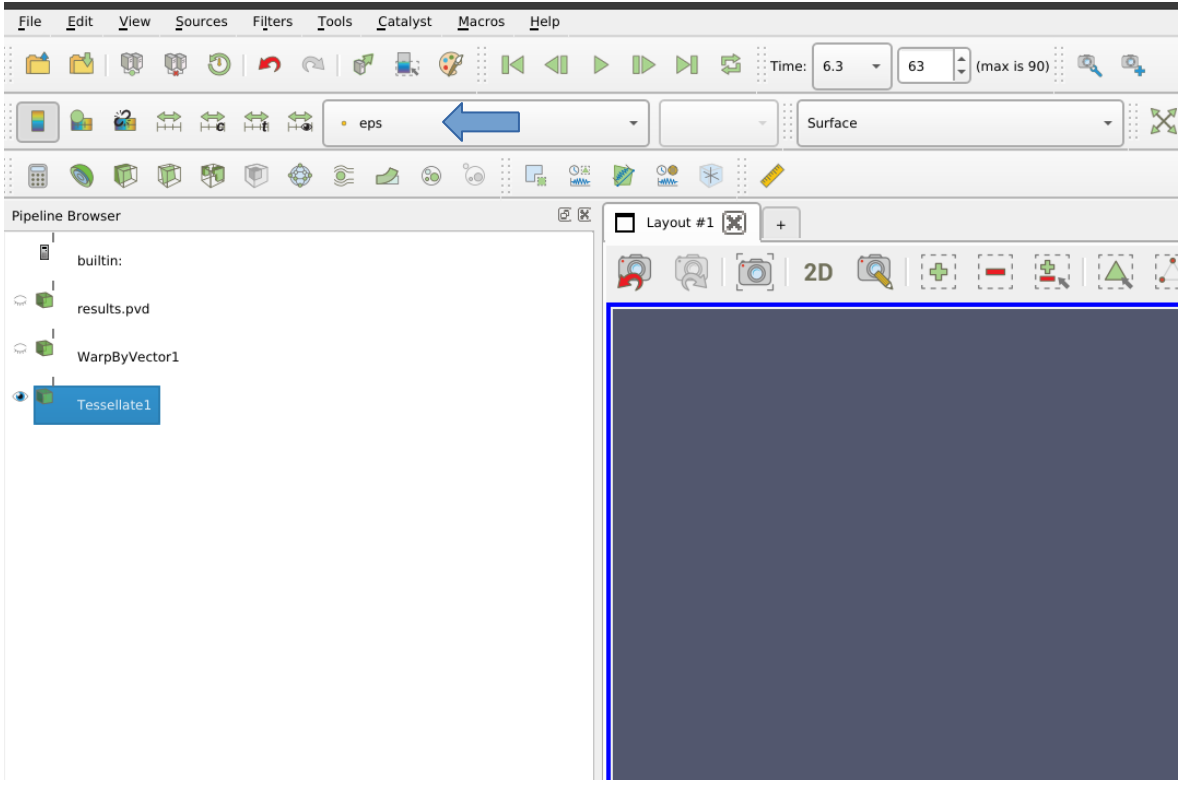


Play Displacments

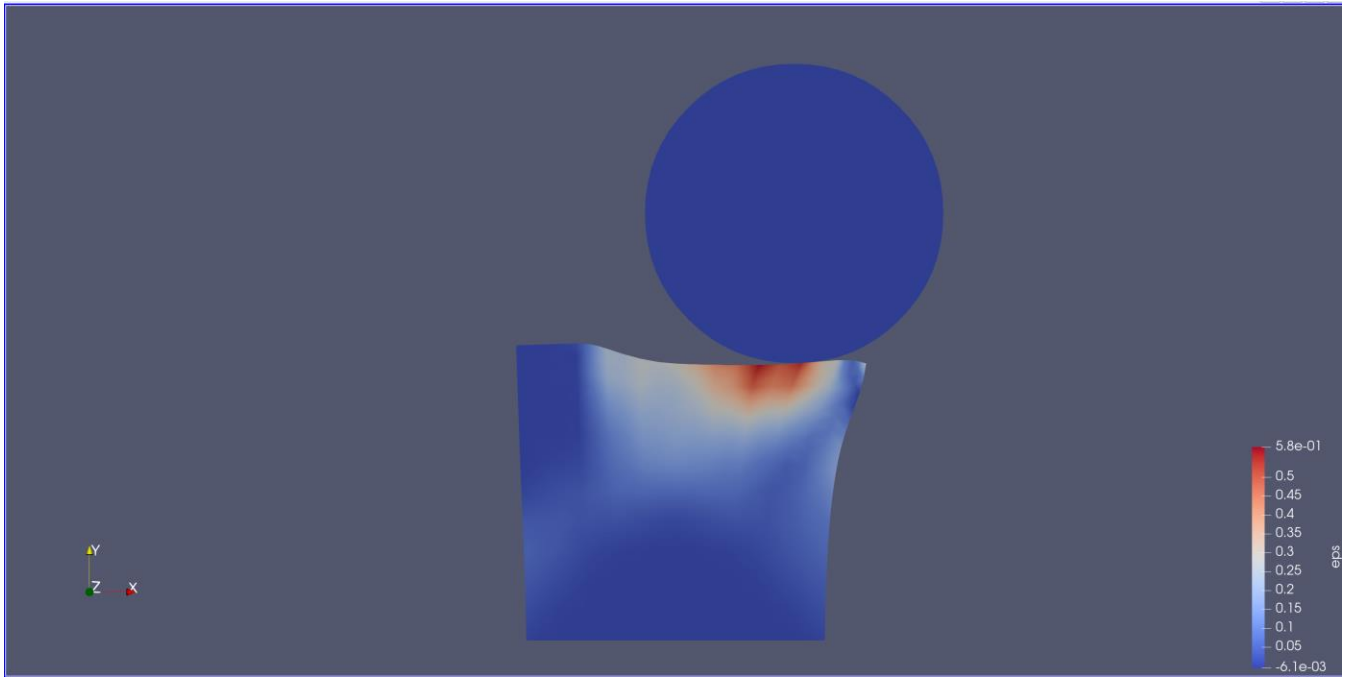


You may need to adjust the color scaling

View eps

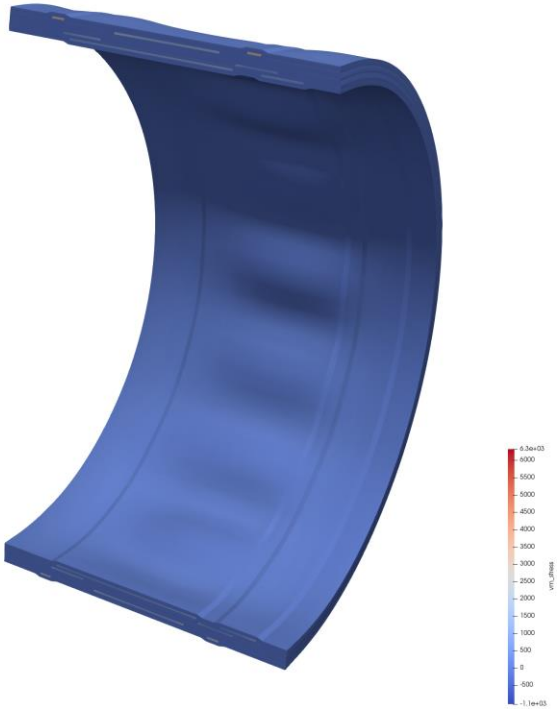

File
Edit
View
Sources
Filters
Tools
Catalyst
Macros
Help
Time:
6.3
63
(max is 90)
eps
Surface
Pipeline Browser
builtin:
results.pvd
WarpByVector1
Tessellate1
Layout #1
+
2D

Play eps



You may need to adjust the color scaling

Problem 3: Flex Cable



This is a specific industry application that has been proven to be difficult to simulate using traditional FEA. It consists of a flexible cable made up of copper traces surrounded by adhesive and encased in Kapton. The simulation has 2 stages: first cable is bent into a U shape and then a heat load is applied.

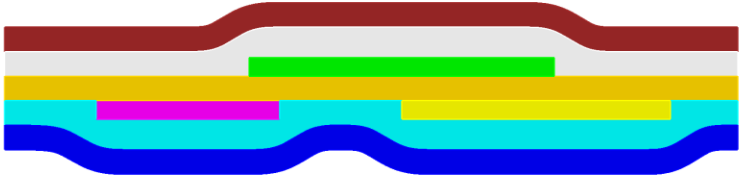
A custom workflow has been created to simulate this problem. The first step is to create a U-spline of the cable cross section. The cross section is then extruded to form a 3D solid and the boundary conditions are applied. The extrusion and application of boundary conditions are automated using a Python script

Learning Objectives

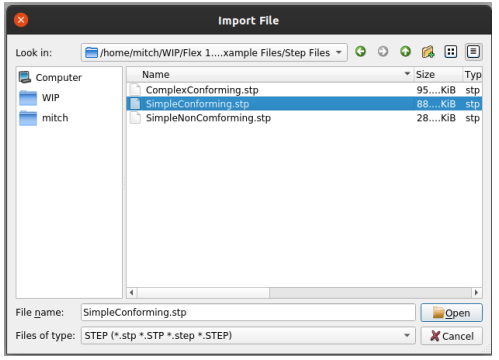
Tutorial participants will learn:

- A custom workflow created for a specific customer application
- How to import multiple cards to add to your simulation

Import geometry



File->Import



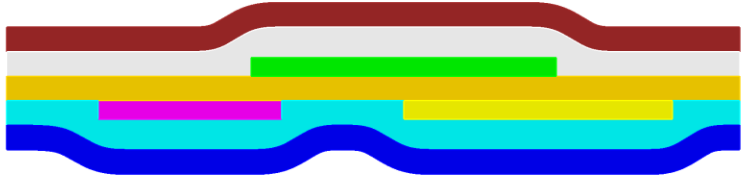
Accept import default options

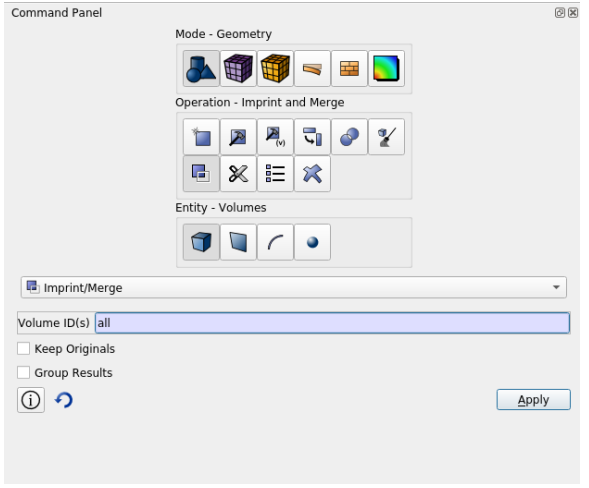
- or-

Import step "<path>" heal

This file is available at https://coreform.com/shortcourse/FlexCable/SimpleConforming.stp

Imprint and merge

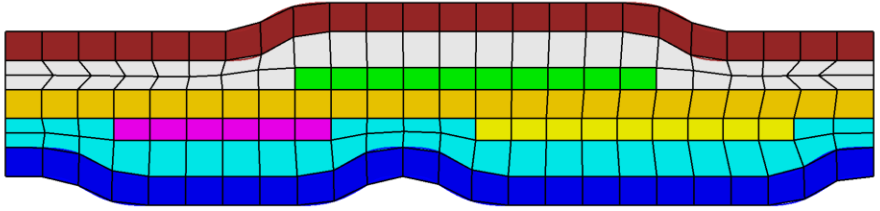


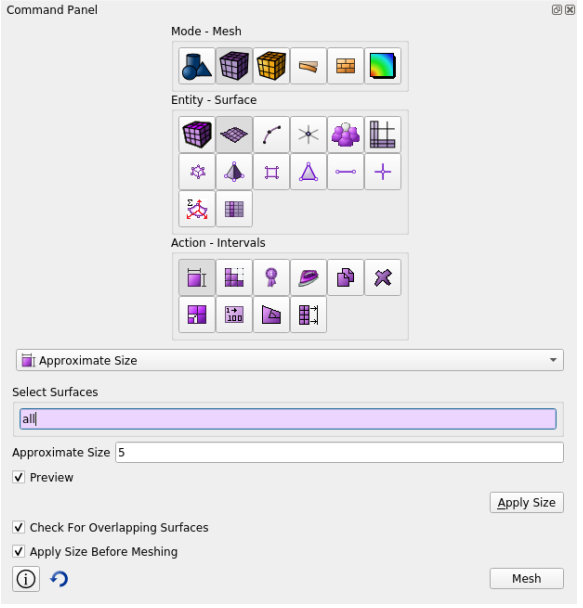


- or -

Imprint all
Merge all

Set mesh size then mesh surface

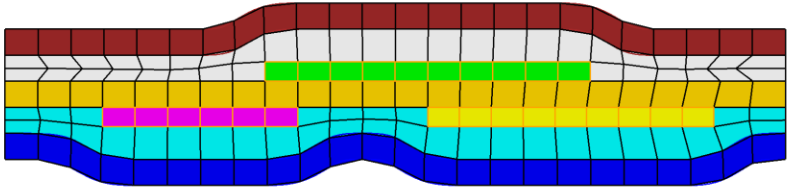


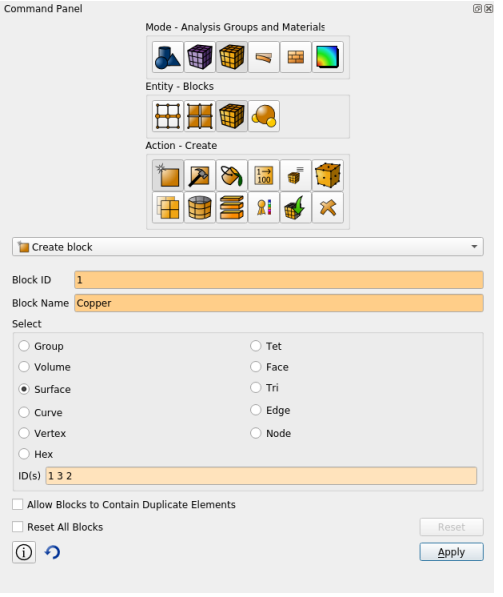


- or -

surface all size 5
Mesh surface all

Setup Trelis Blocks - Copper

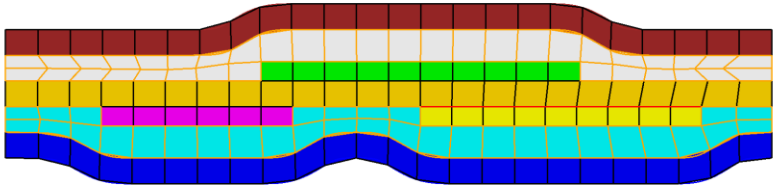


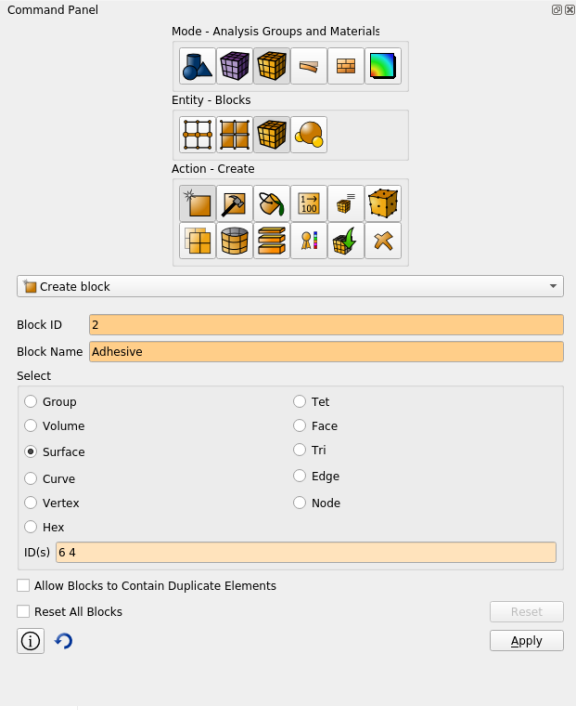


- or -

Block 1 add surface 1 2 3
Block 1 name "Copper"

Setup Trelis Blocks - Adhesive

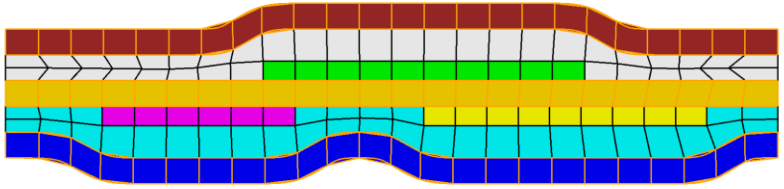


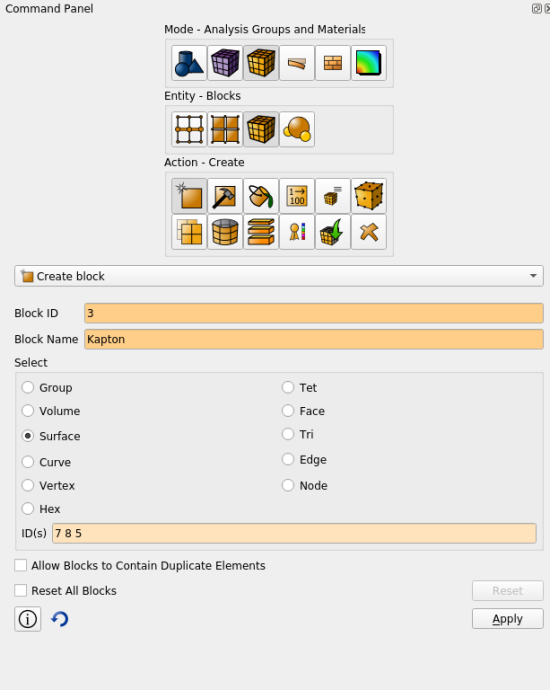


- or -

Block 2 add surface 6 4
Block 2 name "adhesive"

Setup Trelis Blocks - Kapton

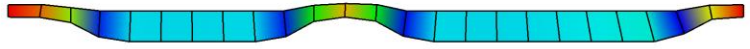


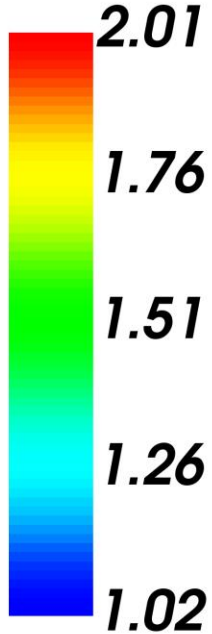


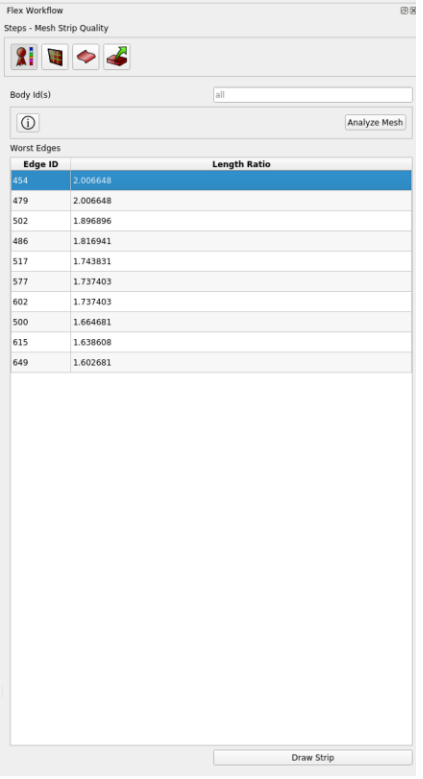
- or -

Block 3 add surface 5 7 8
Block 3 name "kapton"

Check strip quality



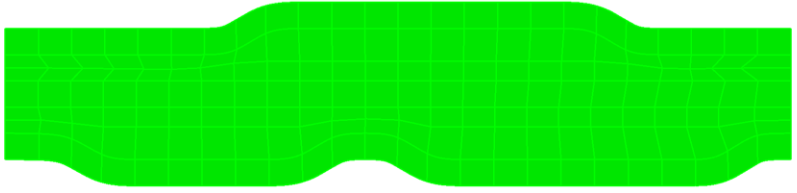


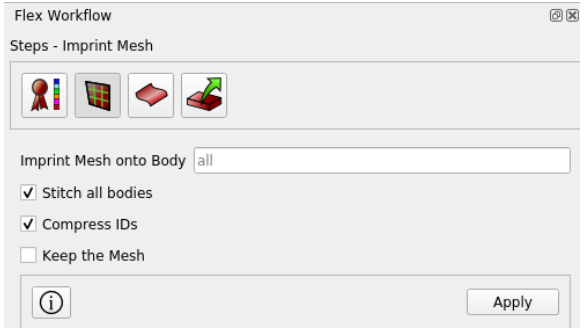


- or -

Uspline strip quality body all

Imprint mesh onto geometry

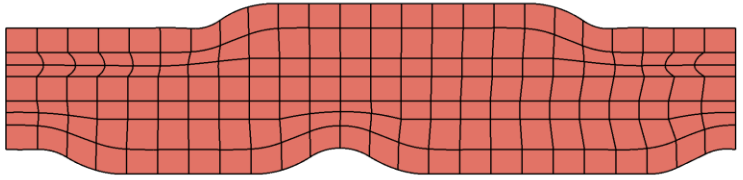


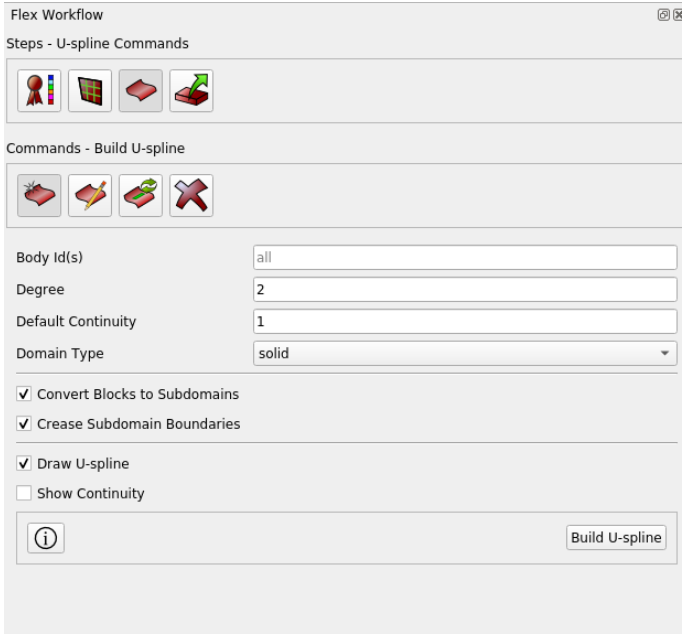


- or -

Imprint mesh onto body all
Stitch body all
compress

Build U-spline

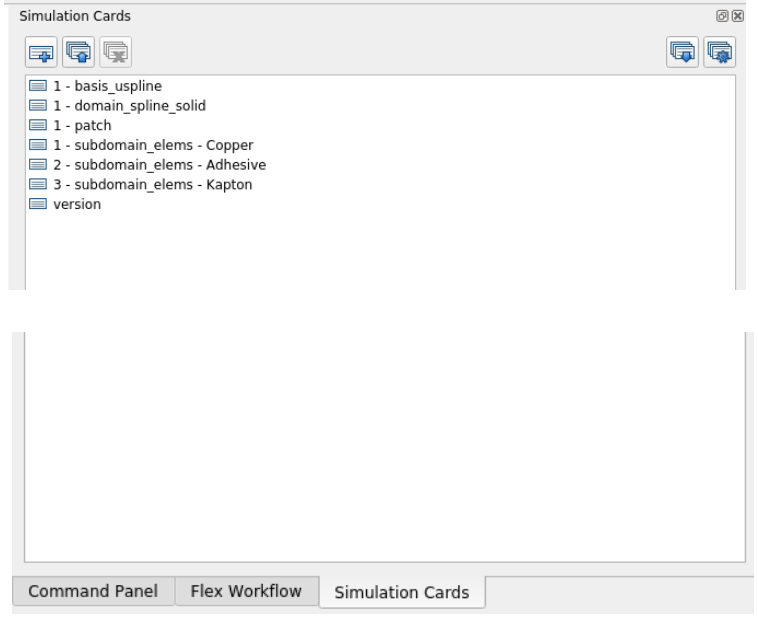




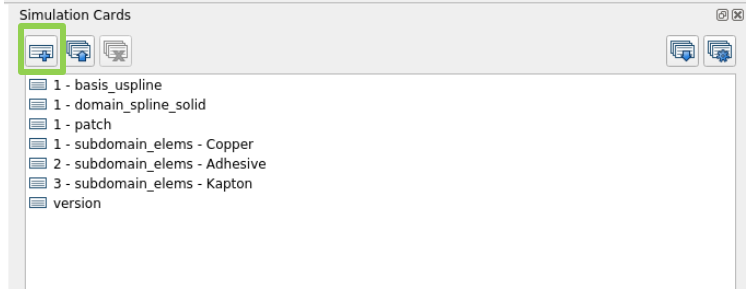
- or -

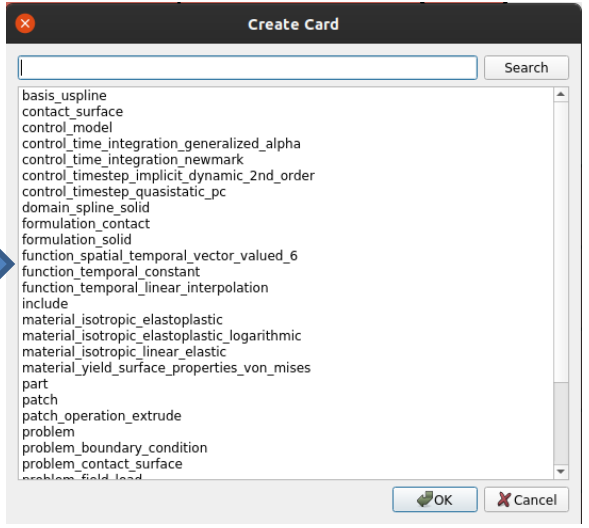
Build uspline body all p 2 c 1 domain "solid" draw

View simulation cards



Create card


Simulation Cards
1 - basis_uspline
1 - domain_spline_solid
1 - patch
1 - subdomain_elems - Copper
2 - subdomain_elems - Adhesive
3 - subdomain_elems - Kapton
version


Create Card
Search
basis_uspline
contact_surface
control_model
control_time_integration_generalized_alpha
control_time_integration_newmark
control_timestep_implicit_dynamic_2nd_order
control_timestep_quasistatic_pc
domain_spline_solid
formulation_contact
formulation_solid
function_spatial_temporal_vector_valued_6
function_temporal_constant
function_temporal_linear_interpolation
include
material_isotropic_elastoplastic
material_isotropic_elastoplastic_logarithmic
material_isotropic_linear_elastic
material_yield_surface_properties_von_mises
part
patch
patch_operation_extrude
problem
problem_boundary_condition
problem_contact_surface
OK
Cancel

Add material for adhesive

function_temporal_linear_interpolation

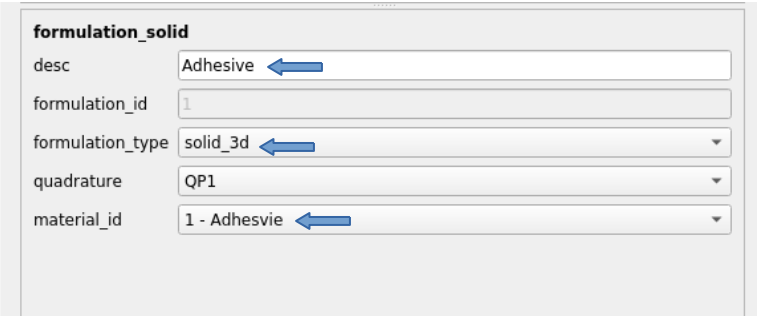
desc	Adhesive Modulus
function_temporal_id	1
birth	70
death	160
tol	1e-10
graph	

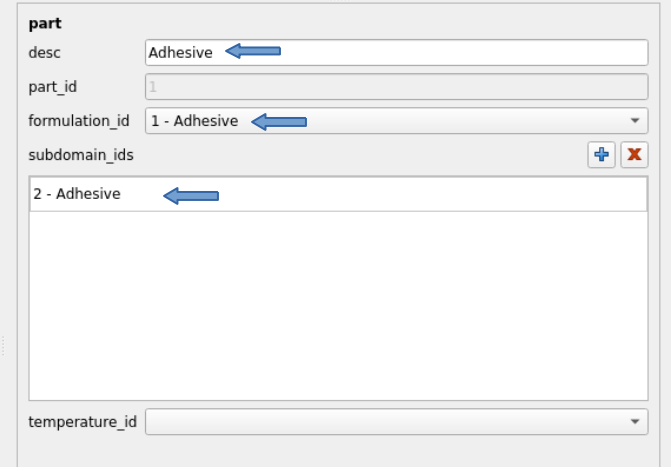
	t	f(t)
1	70	689
2	100	0.689
3	160	0.6

material_isotropic_elastoplastic

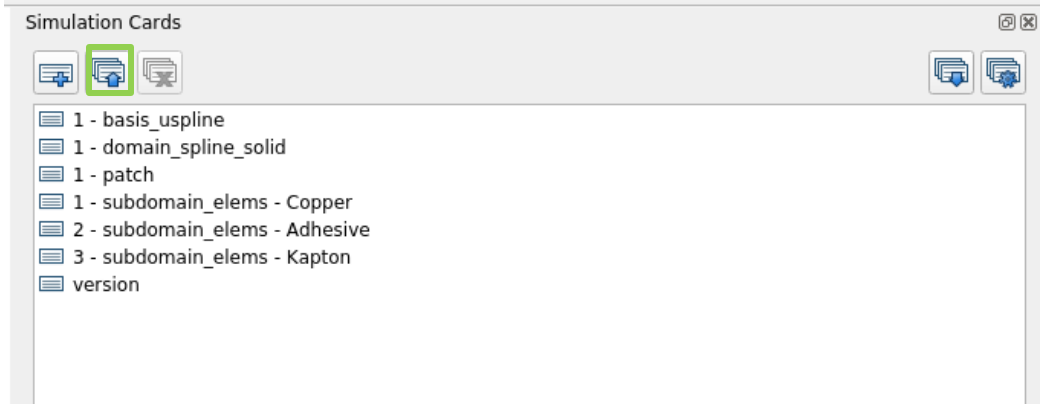
desc	Adhesive
material_id	1
E	
E_time_dependent_function_temporal_id	
E_temperature_dependent_function_temporal_id	1 - Adhesive modulus
nu	0.34
rho	1.07e-06
thermal_expansion	5e-05
None	
function_temporal_id	
K	
yield_stress	
saturation_stress	
saturation_exponent	
plastic_work_threshold	0
effective_plastic_work_measure	plastic_work
yield_type	J2

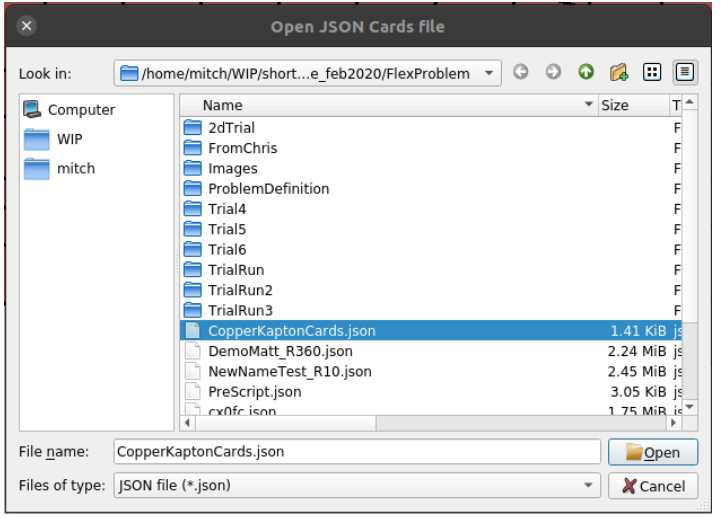
Add adhesive formulation and part





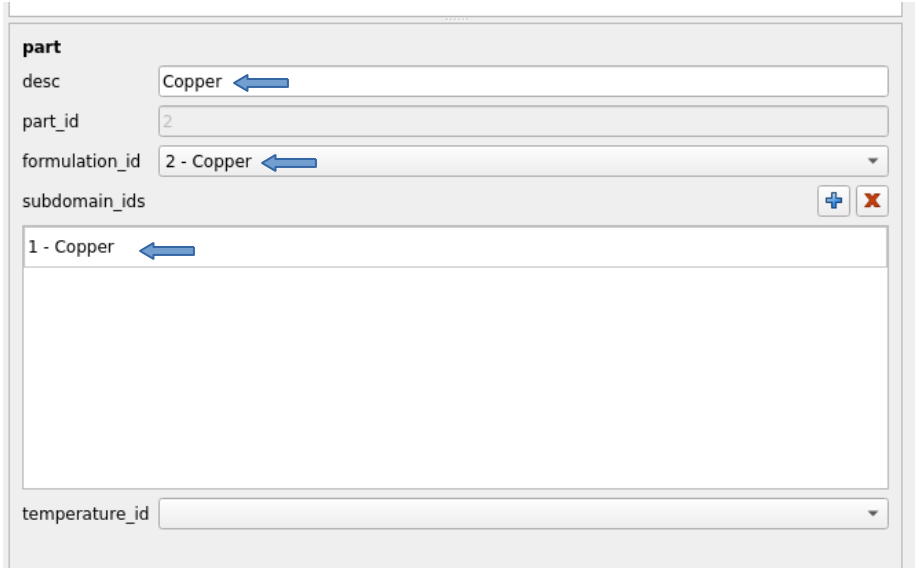
Import cards for copper and kapton

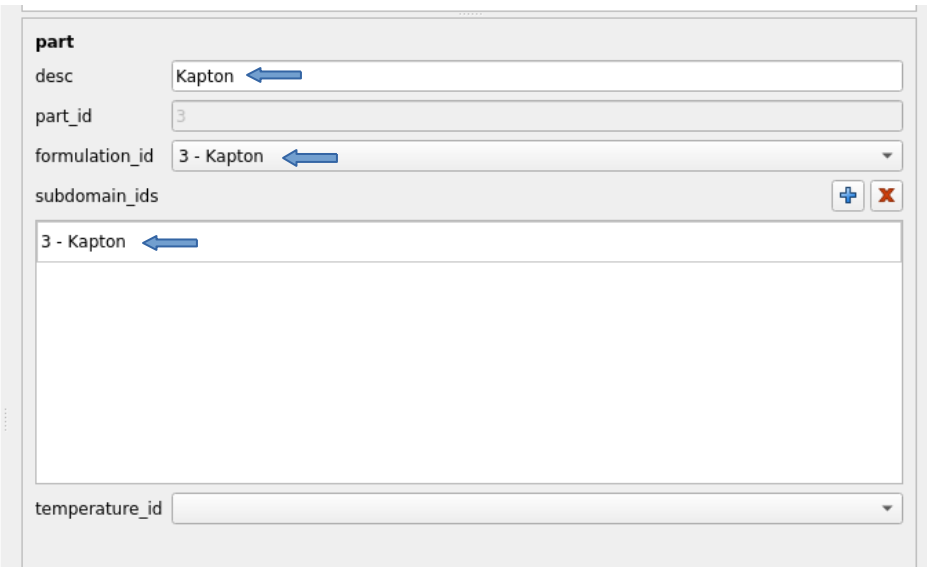




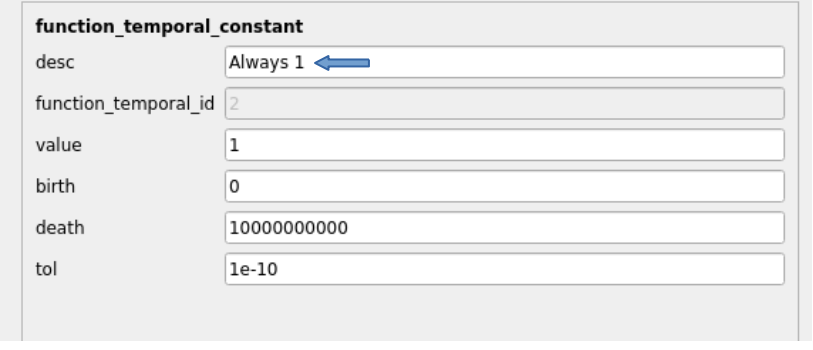
This file is available at https://coreform.com/shortcourse/PressureVessel/CopperKaptonCards.json

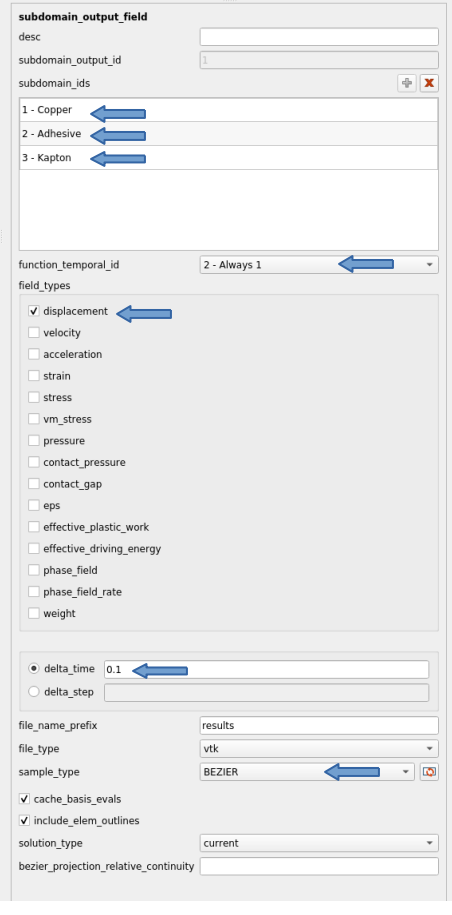
Associate imported parts with correct subdomain



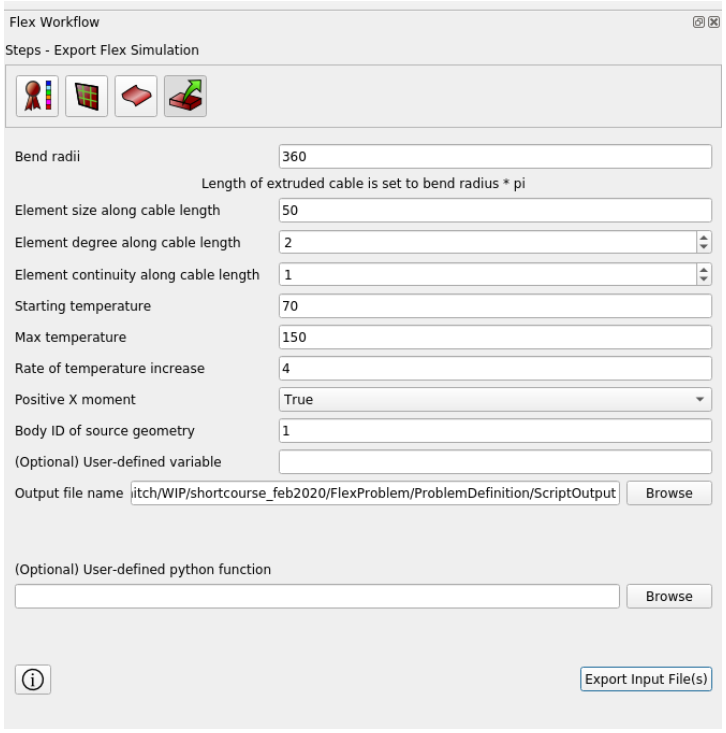


Add output cards





Export Flex simulation file



When the users clicks "Export Input File(s)", the simulation cards and the parameters entered into this dialog are input into a python script that will:

- Add a patch_operation_extrude card that will turn the 2d cross section into a 3d solid model*
- Add boundary conditions
- Add other cards required for the simulation (control_model, problem etc)
- Export this modified solid simulation input file that is ready to run in the solver

*Visualization of 3D U-splines is not currently supported in Trelis, so it is not possible to view the solid flex cable model

Import Flex simulation cards