



coreform

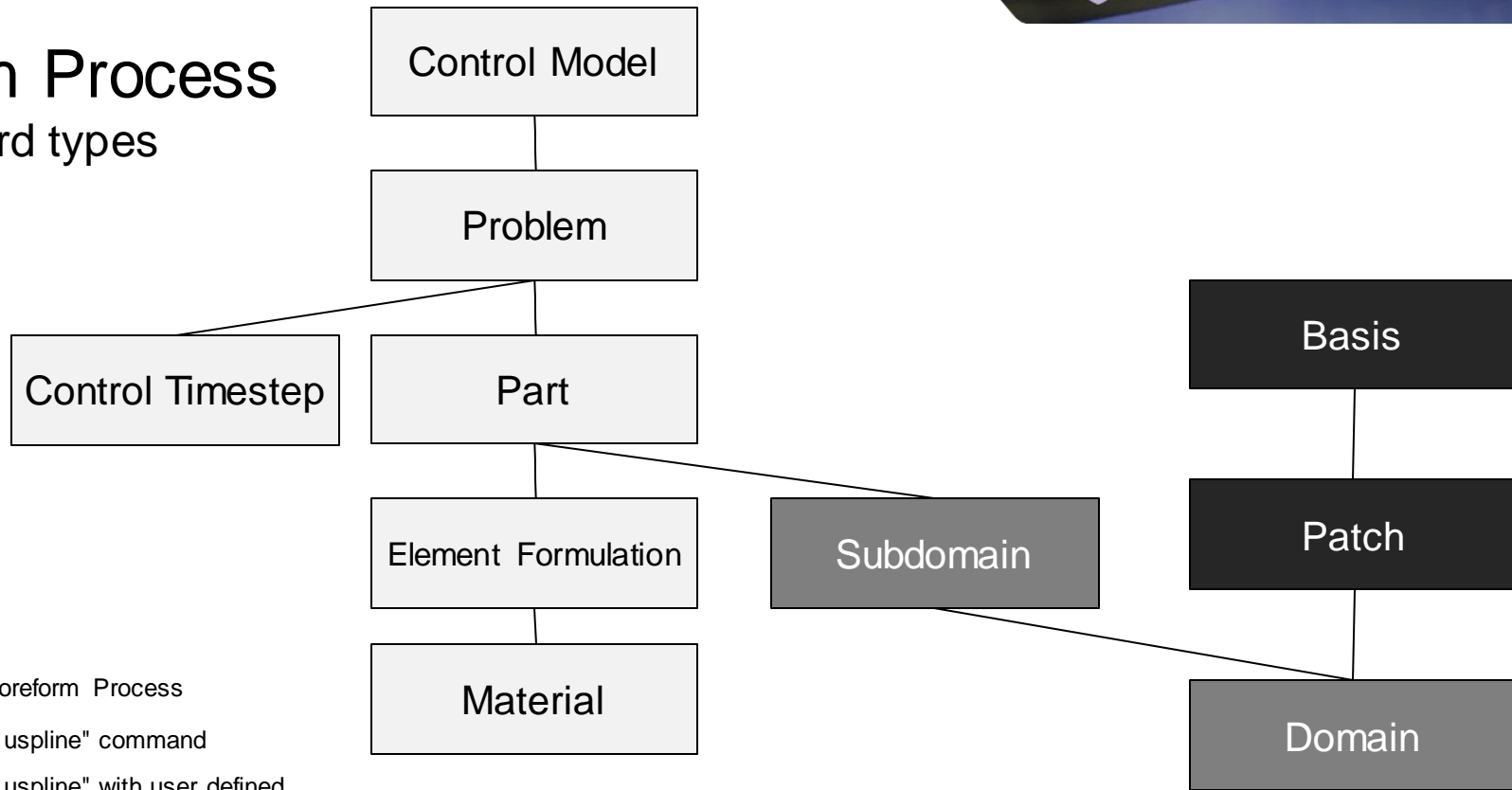
**Better simulation through
better geometry**




Short Course Example Problems

February 20, 2020
Provo, UT

Coreform Process

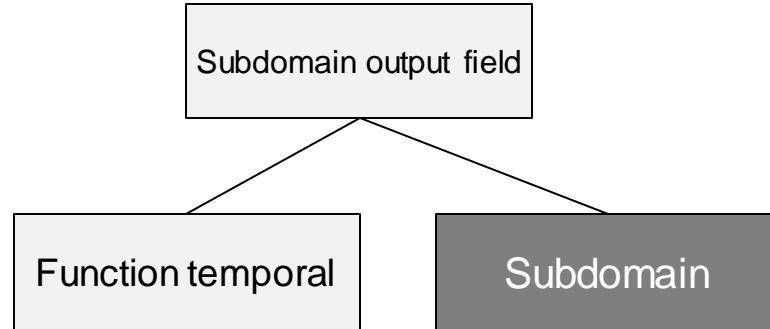
Required card types






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

Coreform Process

Output cards

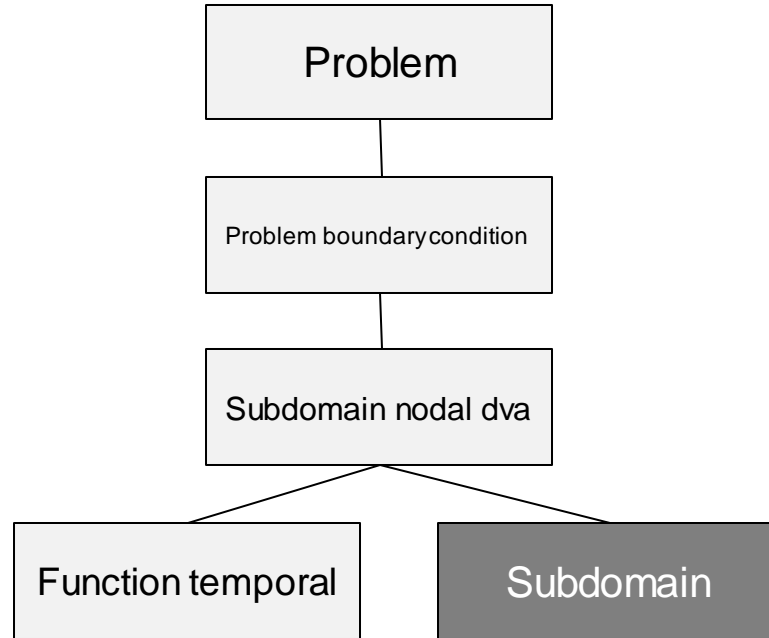





-  User-defined in Coreform Process
-  Created by "build uspline" command
-  Created by "build uspline" with user defined setting

Coreform Process

Displacement Boundary

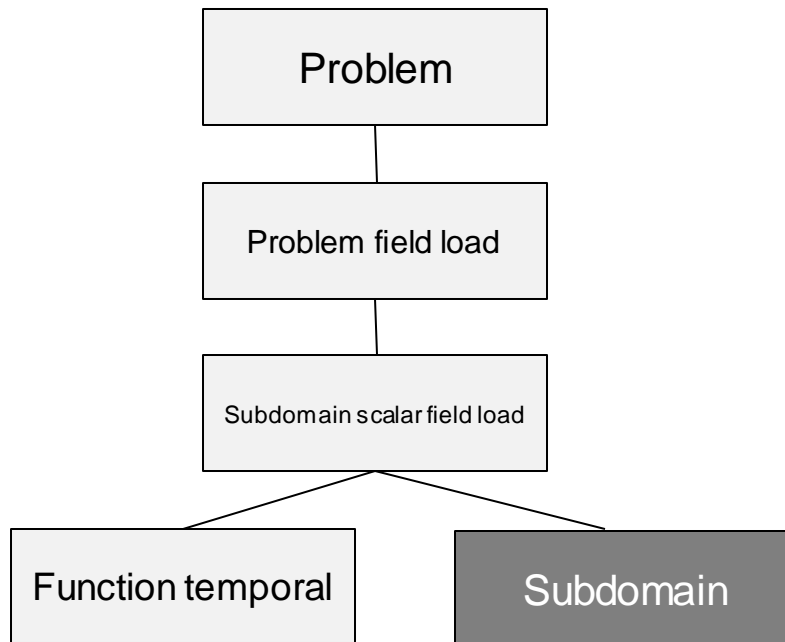
Condition cards






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

Coreform Process

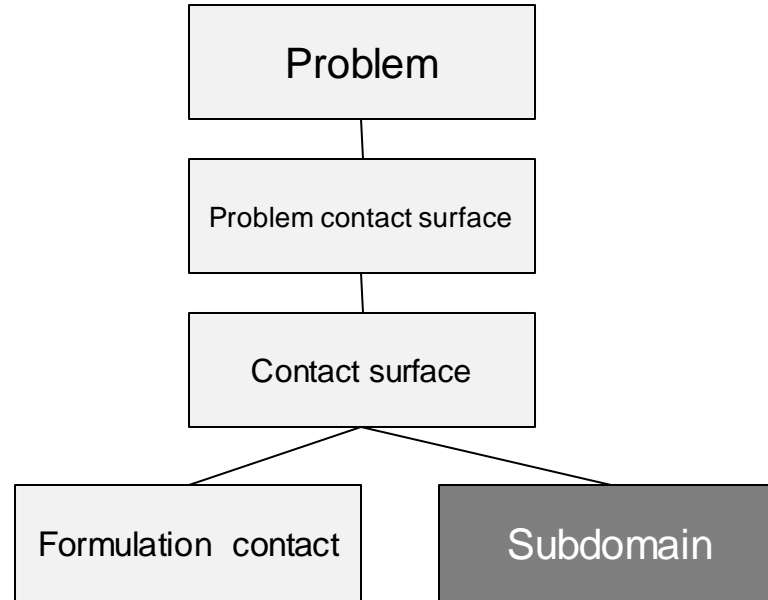
Pressure load cards






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

Coreform Process

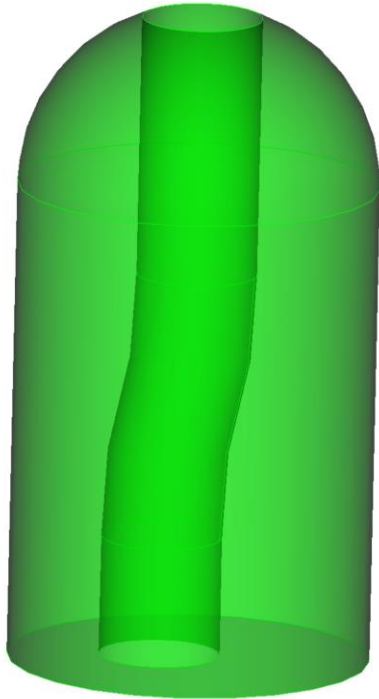
Contact cards



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

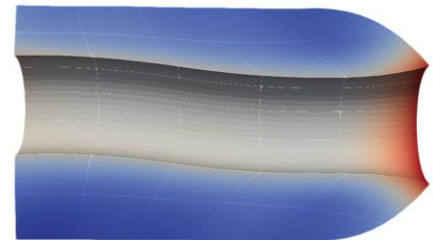
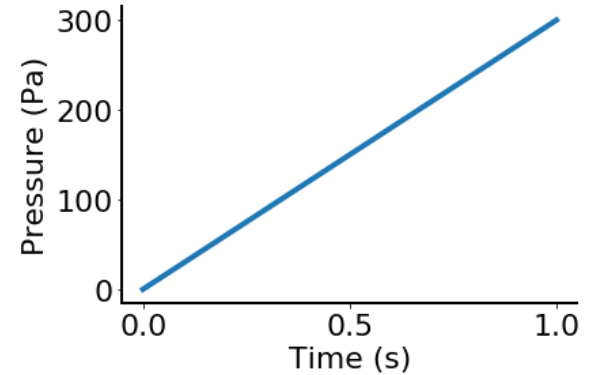
Problem 1: Simple unstructured solid simulation

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



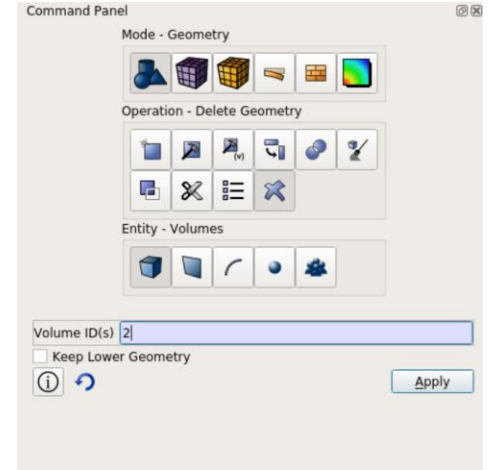
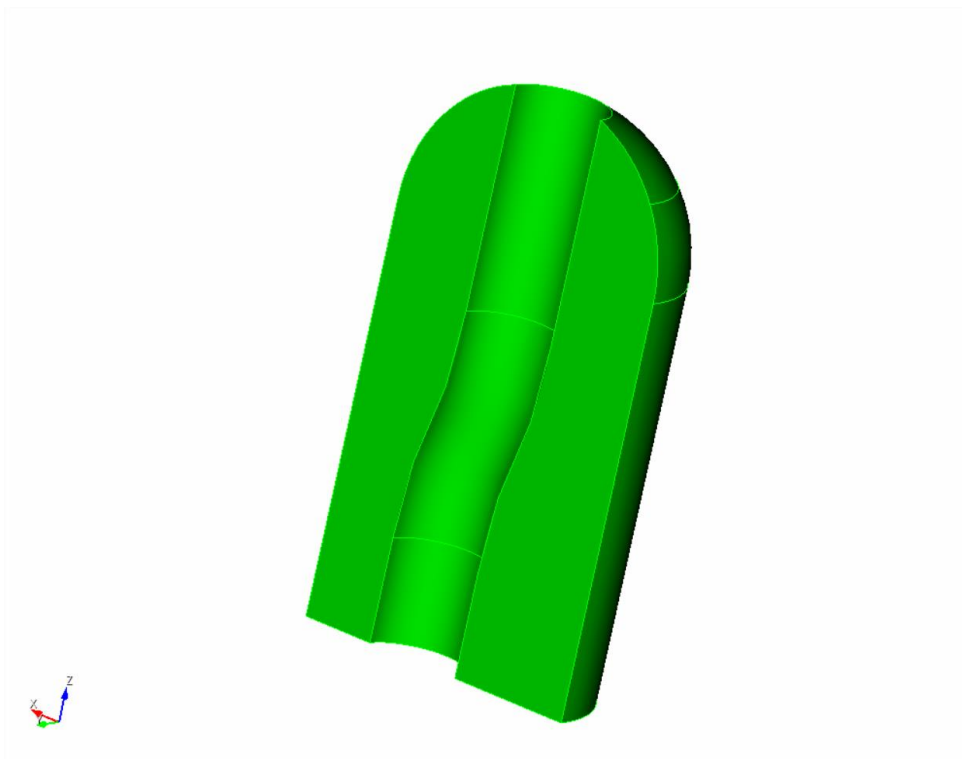
Webcut part on ZX plane



- or -

webcut volume 1 with plane yplane

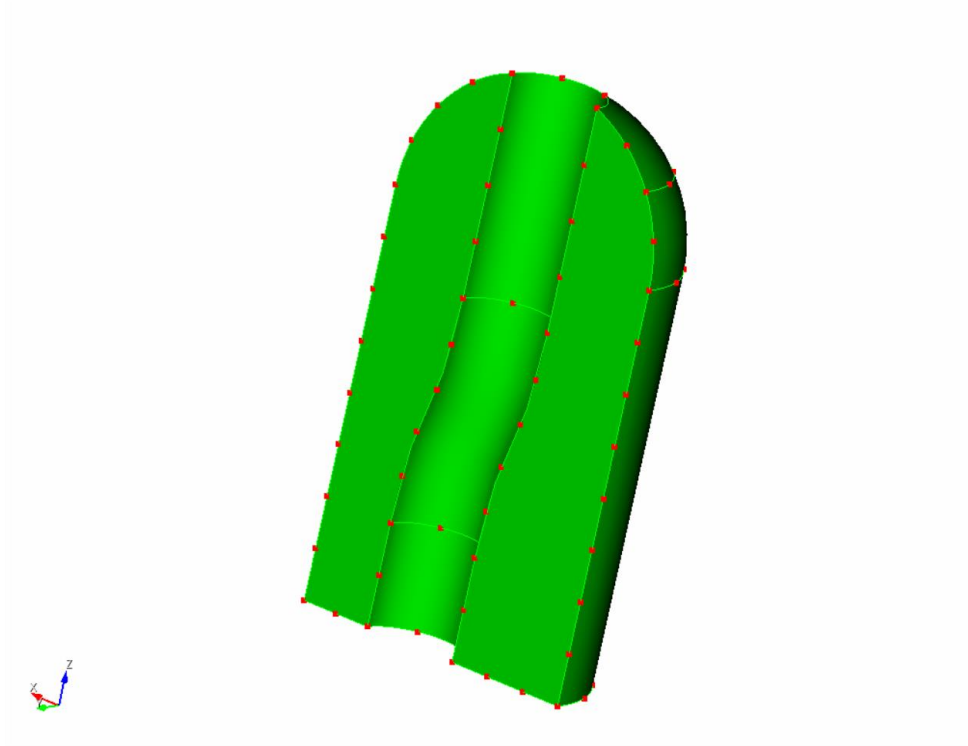
Delete half of the part



- or -

delete volume 2

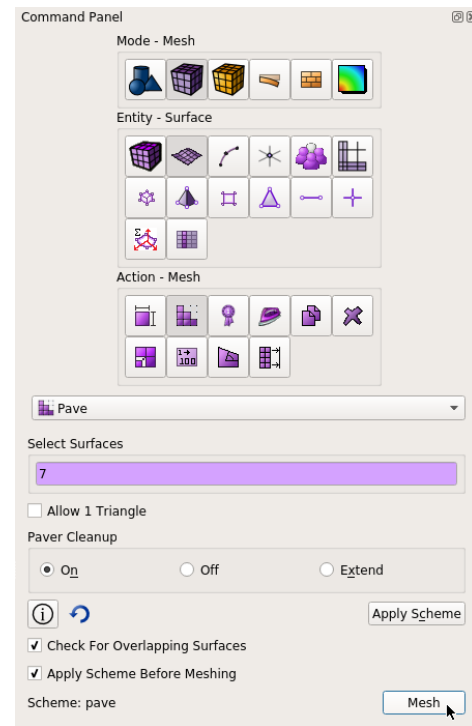
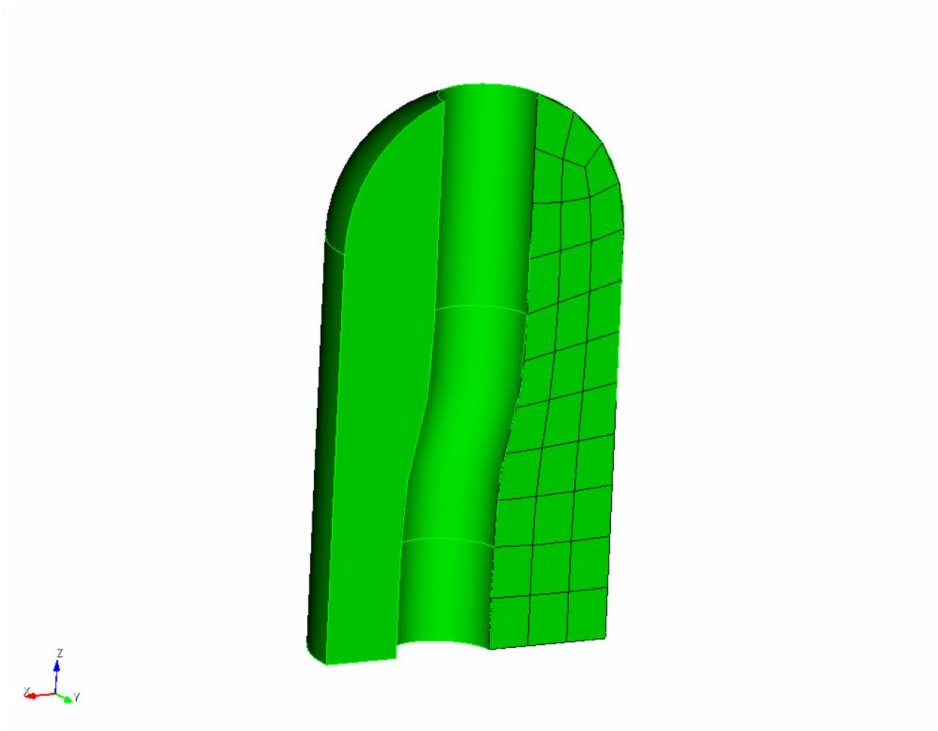
Set mesh element size



- or -

volume 1 size 1

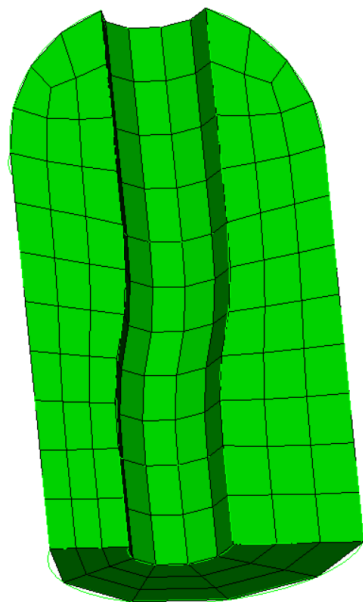
Mesh face



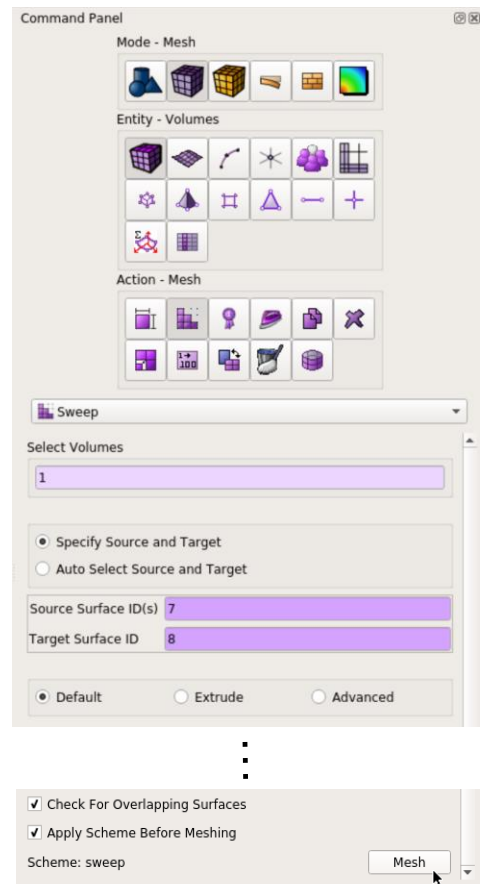
- or -

surface 7 scheme pave
mesh surface 7

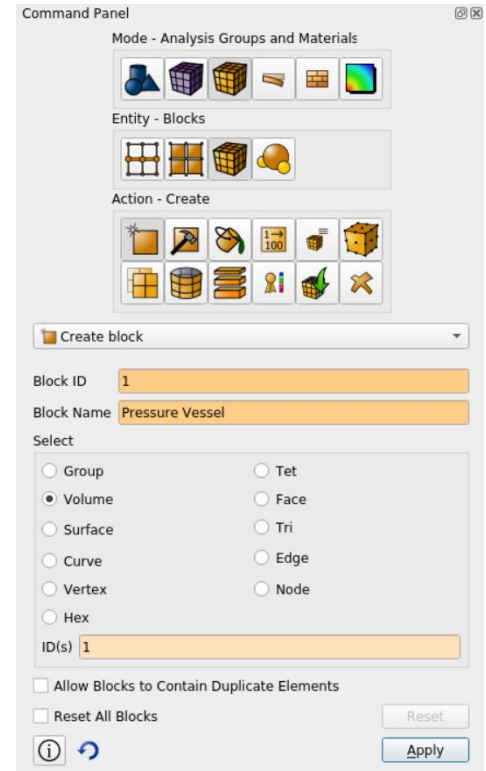
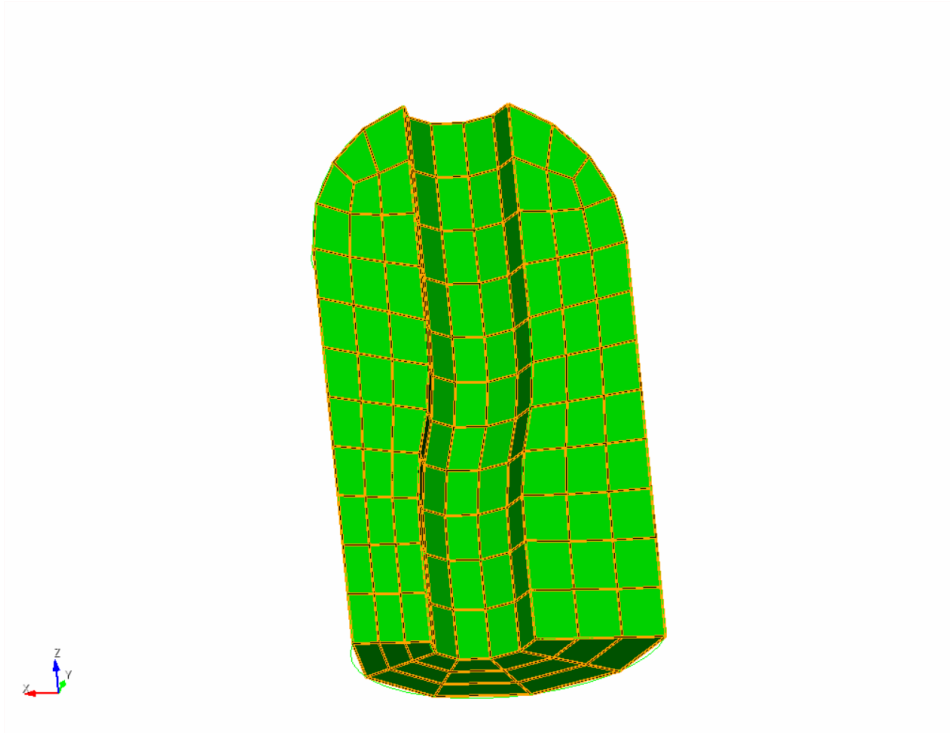
Mesh volume as swept surface



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



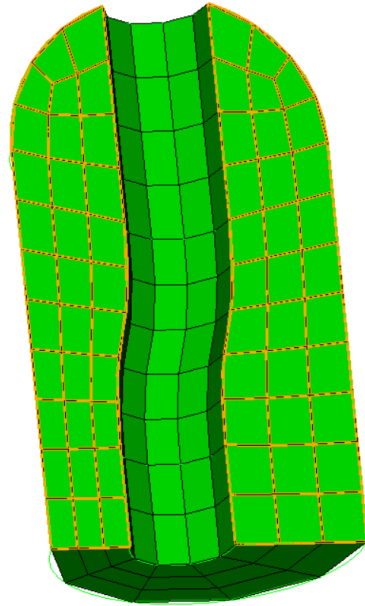
Create block



- or -

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



Create sidesets – symmetry boundary



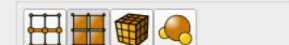
```
sideset 1 add surface 7 8  
sideset 1 name "Symmetry Boundary"
```

Command Panel ⊞ ⓧ

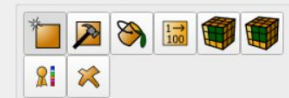
Mode - Analysis Groups and Materials



Entity - Sidesets



Action - Create sideset



Sideset ID

Sideset Name

Select

Surface Face
 Curve Edge
 Group Tri

ID(s)

With Respect To

wrt Surface wrt Hex
 wrt Volume wrt Tet
 wrt Face wrt Tri

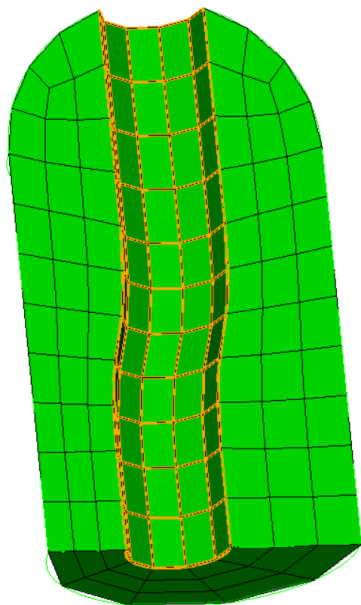
Surface ID(s)

Direction

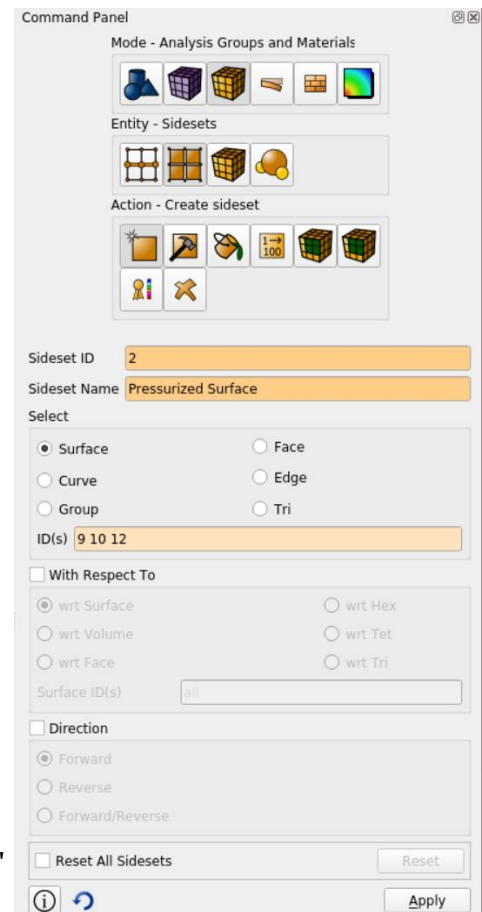
Forward Reverse
 Forward/Reverse

Reset All Sidesets

Create sidesets – pressurized surface



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



Command Panel

Mode - Analysis Groups and Materials

Entity - Sidesets

Action - Create sideset

Sideset ID: 2

Sideset Name: Pressurized Surface

Select

- Surface
- Face
- Curve
- Edge
- Group
- Tri

ID(s): 9 10 12

With Respect To

- wrt Surface
- wrt Hex
- wrt Volume
- wrt Tet
- wrt Face
- wrt Tri

Surface ID(s): all

Direction

- Forward
- Reverse
- Forward/Reverse

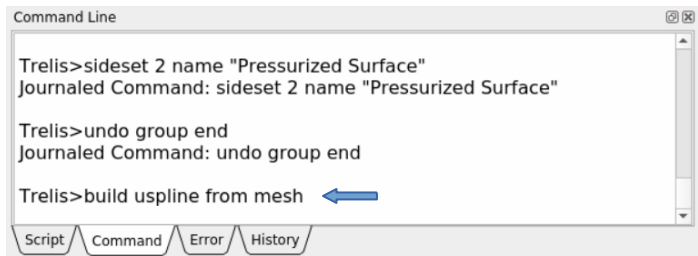
Reset All Sidesets

Reset

Apply

Generate and view simulation geometry cards

The simulation cards tab should now look like this:



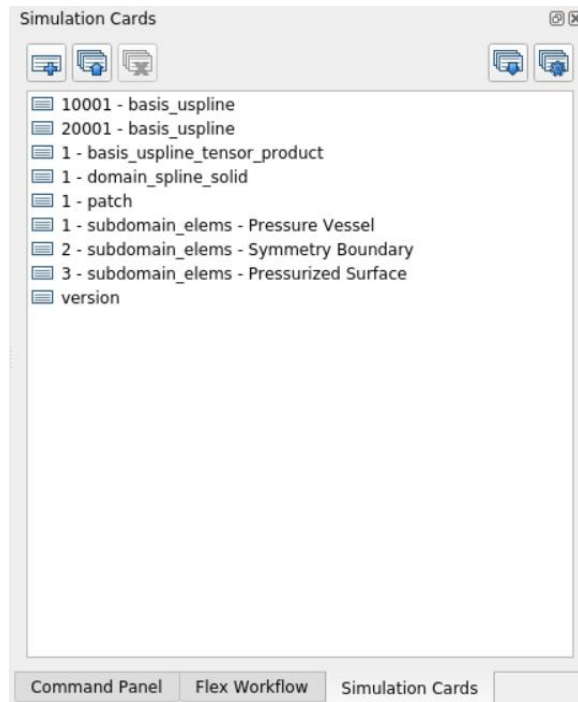
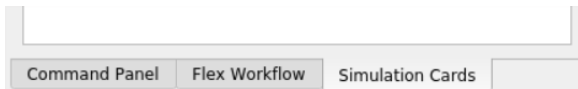
```
Command Line
Trelis> sideset 2 name "Pressurized Surface"
Journaled Command: sideset 2 name "Pressurized Surface"

Trelis> undo group end
Journaled Command: undo group end

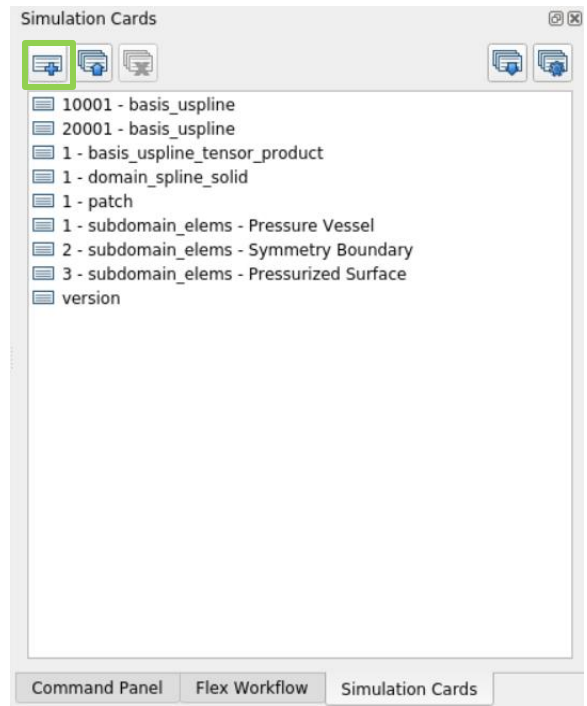
Trelis> build uspline from mesh
```

Build is finished when Trelis> prompt returns.

Select "Simulation Cards" tab in bottom right



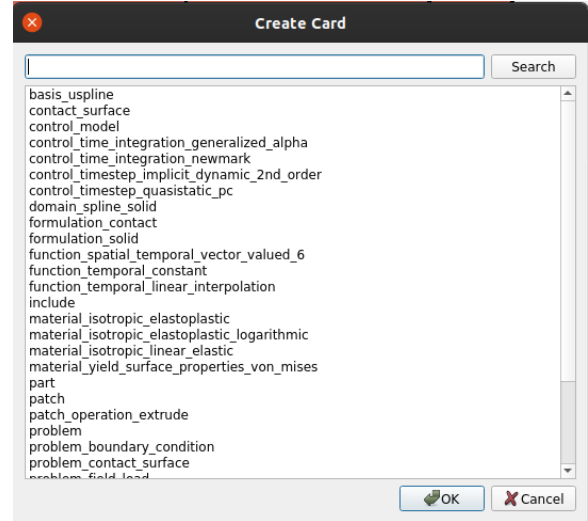
Adding Cards



The 'Simulation Cards' window shows a list of simulation cards. The 'Add' icon (a plus sign in a square) is highlighted with a green box. The list includes:

- 10001 - basis_uspline
- 20001 - basis_uspline
- 1 - basis_uspline_tensor_product
- 1 - domain_spline_solid
- 1 - patch
- 1 - subdomain_elems - Pressure Vessel
- 2 - subdomain_elems - Symmetry Boundary
- 3 - subdomain_elems - Pressurized Surface
- version

At the bottom, there are tabs for 'Command Panel', 'Flex Workflow', and 'Simulation Cards'.



The 'Create Card' dialog window shows a search bar and a list of available simulation cards:

- 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
- problem_field_load

Buttons for 'OK' and 'Cancel' are at the bottom right.

Add material and formulation cards

material_isotropic_linear_elastic

desc

material_id

E


E_time_dependent_function_temporal_id

E_temperature_dependent_function_temporal_id

nu

rho

thermal_expansion



formulation_solid


desc

formulation_id

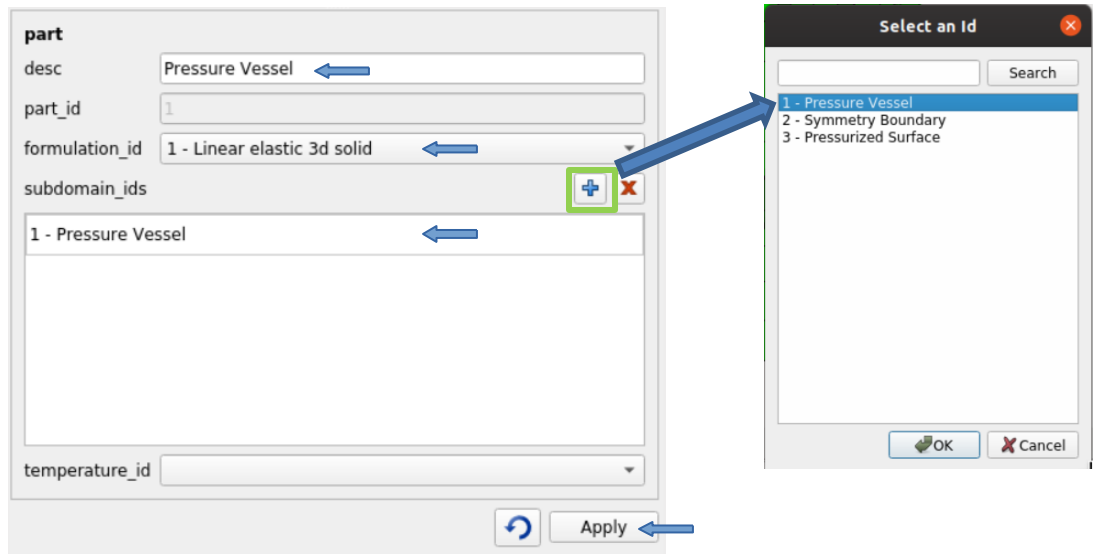
formulation_type

quadrature

material_id



Add part card



The image shows a software interface for configuring a 'part' card. The main window has the following fields:

- part** (header)
- desc**: Pressure Vessel
- part_id**: 1
- formulation_id**: 1 - Linear elastic 3d solid
- subdomain_ids**: 1 - Pressure Vessel
- temperature_id**: (empty)

A green box highlights a '+' icon in the subdomain_ids field, which is linked by a blue arrow to a 'Select an Id' dialog box. The dialog box contains a search field and a list of options:

- 1 - Pressure Vessel (highlighted)
- 2 - Symmetry Boundary
- 3 - Pressurized Surface

Buttons for 'OK' and 'Cancel' are at the bottom of the dialog. In the main window, an 'Apply' button is also highlighted with a blue arrow.

Add timestep cards

control_time_integration_newmark

desc

control_time_integration_id

beta

gamma

control_timestep_implicit_dynamic_2nd_order

desc

control_timestep_id

control_time_integration_id

predictor_type

max_corrector_step_n

newton_tol_abs

newton_tol_rel

delta_tol_abs

delta_tol_rel

line_search

line_search_tol

Add problem and control_model cards

problem

desc

problem_id

part_ids

control_timestep_id

coupled_problems

control_linear_solver

control_model

desc

control_time

initial_time_step

termination_time

adaptive_timestep

iteration_optimal

iteration_window

growth_factor

reduction_factor

delta_t_min

delta_t_max

control_problem

enable_parent_basis

Add temporal functions

function_temporal_constant

desc


function_temporal_id

value

birth

death

tol



function_temporal_linear_interpolation



desc





function_temporal_id


birth

death

tol

graph  

	t		f(t)
1	0		0 
2	1		1 



Add symmetry boundary condition

subdomain_nodal_dva

desc: Symmetry Boundary Condition

subdomain_nodal_value_id: 1

subdomain_id: 2 - Symmetry Boundary

dva_type: DISPLACEMENT

UX: 0

UY: 0

UZ: 0

RX: 0

RY: 0

RZ: 0

function_temporal_id: 1 - Always and Forever

function_spatial_temporal_id: [empty]

Apply

problem_boundary_condition

desc: Associate BCs with problem

problem_id: 1 - problem

subdomain_nodal_value_ids: [+] [X]

1 - Symmetry Boundary Condition

Apply

Add pressure load

subdomain_scalar_field_load

desc: Pressure on inside face

subdomain_field_value_id: 1

subdomain_id: 3 - Pressurized Surface

load_type: pressure

magnitude: 300

domain_type: reference

function_temporal_id: 2 - Pressure variation with time

function_spatial_temporal_id:

Apply

problem_field_load

desc: Associate load with problem

problem_id: 1 - problem

subdomain_field_value_ids: + X

1 - subdomain_scalar_field_load

Apply

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

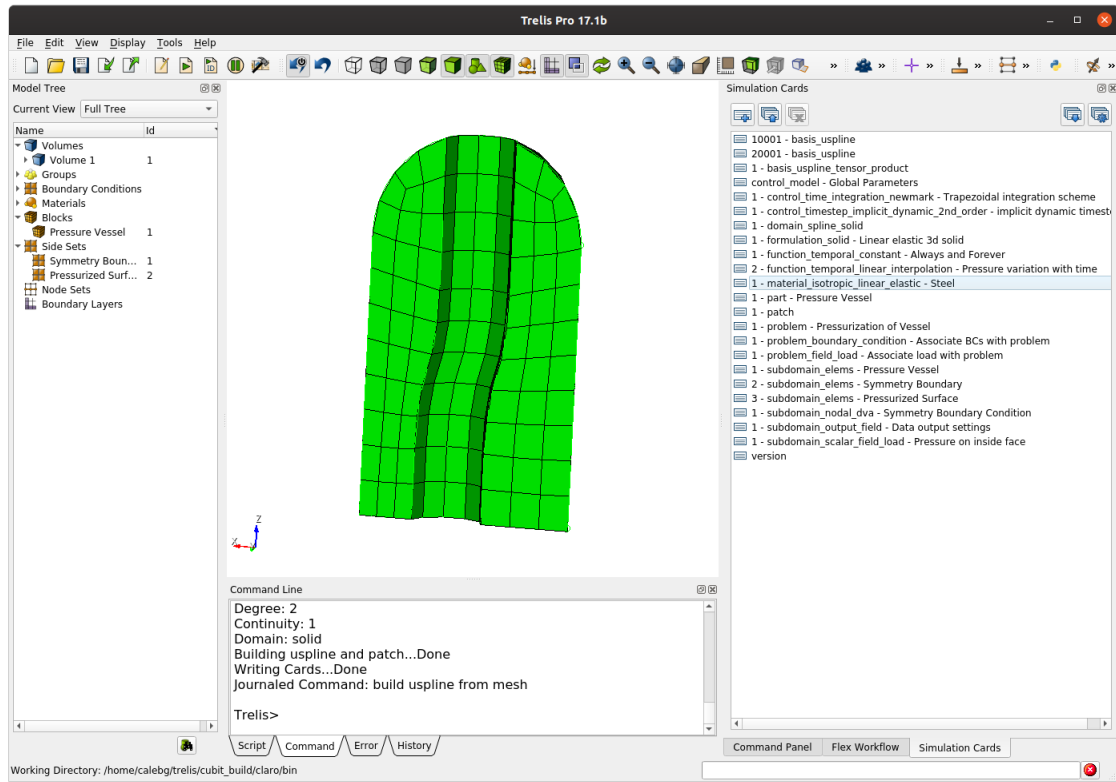
solution_type current

bezier_projection_relative_continuity

Apply

Input finished

Your window should now look like this:



Run Simulation

Simulation Cards

- 10001 - basis_uspline
- 20001 - basis_uspline
- 1 - basis_uspline_tensor_product
- control_model - Global Parameters
- 1 - control_time_integration_newmark - Trapezoidal integration scheme
- 1 - control_timestep_implicit_dynamic_2nd_order - implicit dynamic timestep
- 1 - domain_spline_solid
- 1 - formulation_solid - Linear elastic 3d solid
- 1 - function_temporal_constant - Always and Forever
- 2 - function_temporal_linear_interpolation - Pressure variation with time
- 1 - material_isotropic_linear_elastic - Steel
- 1 - part - Pressure Vessel
- 1 - patch
- 1 - problem - Pressurization of Vessel
- 1 - problem_boundary_condition - Associate BCs with problem
- 1 - problem_field_load - Associate load with problem
- 1 - subdomain_elems - Pressure Vessel
- 2 - subdomain_elems - Symmetry Boundary
- 3 - subdomain_elems - Pressurized Surface
- 1 - subdomain_nodal_dva - Symmetry Boundary Condition
- 1 - subdomain_output_field - Data output settings
- 1 - subdomain_scalar_field_load - Pressure on inside face
- version

Save cards and run analysis

Look in: /home/calebg/ShortCourse/PressureVessel

Computer
calebg

File name: PressureVesselSim Save

Files of type: *.json Cancel

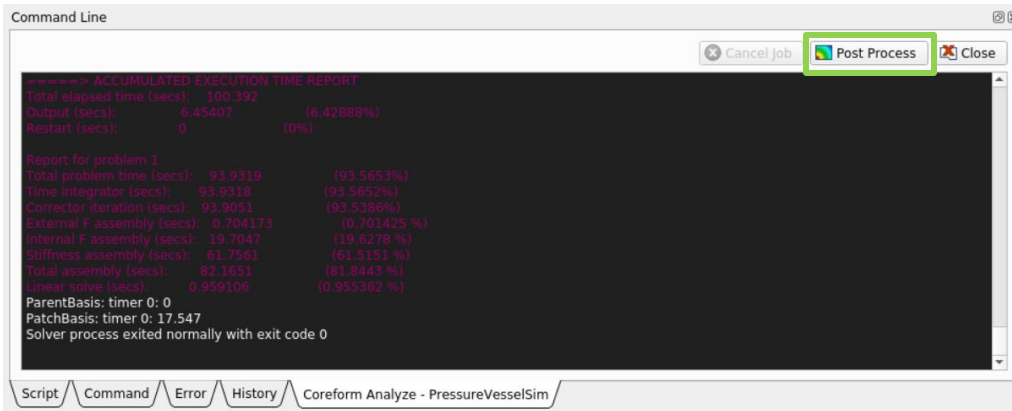
Command Line

Cancel Job Post-Process Close

```
#####>>> Abs Res. norm: 114.883 tol: 1e-32
#####>>> Rel Res. norm: 1 tol: 1e-06
#####>>> Assembling the stiffness matrix...
#####>>> Solving the linear system...
#####>>> Starting corrective Step 2
#####>>> Problem 1
#####>>> Current time 0.1
#####>>> Computing external force...
#####>>> Computing internal force...
#####>>> Checking residual for convergence
#####>>> Abs Res. norm: 0.0267675 tol: 1e-32
#####>>> Rel Res. norm: 0.000232999 tol: 1e-06
#####>>> Abs Delta norm: 0.0369541 tol: 1e-06
#####>>> Rel Delta norm: 1 tol: 0.001
#####>>> Assembling the stiffness matrix...
```

Script Command Error History Coreform Analyze - PressureVesselSim

View Results



Command Line

Cancel Job Post Process Close

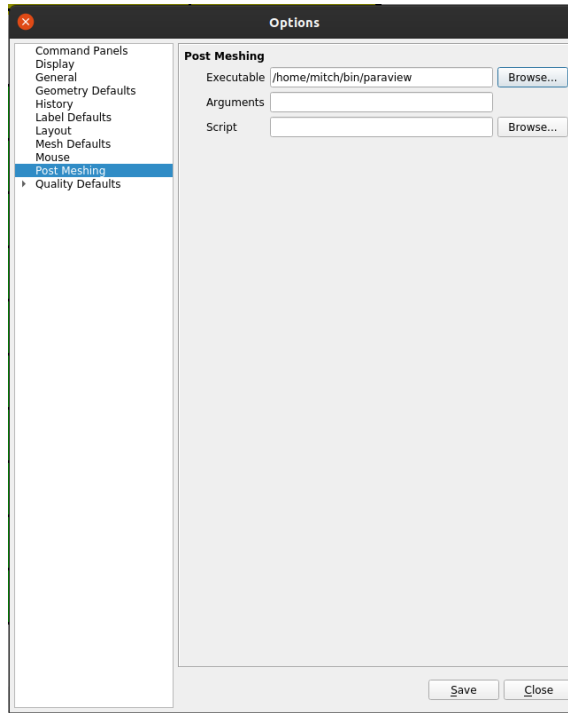
```
-----> ACCUMULATED EXECUTION TIME REPORT
Total elapsed time (secs): 100.392
Output (secs): 6.45407 (6.42888%)
Restart (secs): 0 (0%)

Report for problem 1
Total problem time (secs): 93.9319 (93.5653%)
Time integrator (secs): 93.9318 (93.5652%)
Corrector iteration (secs): 93.9051 (93.5386%)
External F assembly (secs): 0.704173 (0.701425 %)
Internal F assembly (secs): 19.7047 (19.6278 %)
Stiffness assembly (secs): 61.7561 (61.5151 %)
Total assembly (secs): 82.1651 (81.8443 %)
Linear solve (secs): 0.959106 (0.955362 %)

ParentBasis: timer 0: 0
PatchBasis: timer 0: 17.547
Solver process exited normally with exit code 0
```

Script / Command / Error / History / Coreform Analyze - PressureVesselSim

Set Post Process Executable



Tools -> Options -> Post Meshing

View Results

Pipeline Browser

- builtin:
 - results.pvd

Properties Information

Properties

Apply Reset Delete ?

Search ... (use Esc to clear text)

Properties (results.pvd)

- Point/Cell/Column Array Status
 - displacement

Display

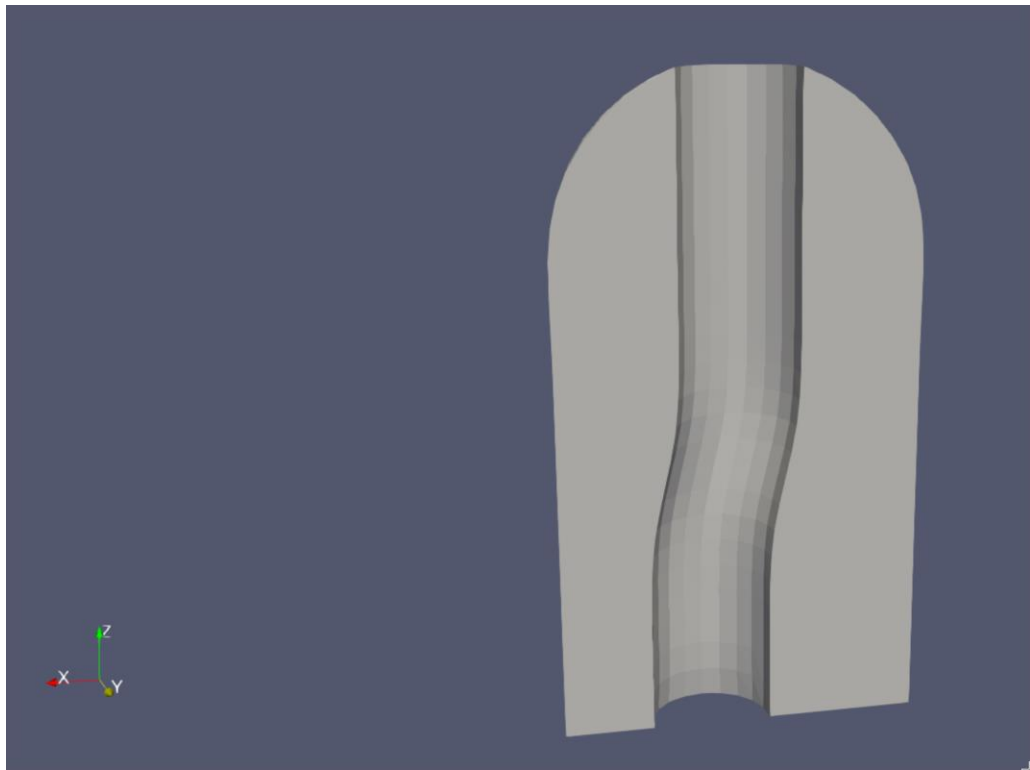
View (Render View)

Axes Grid Edit

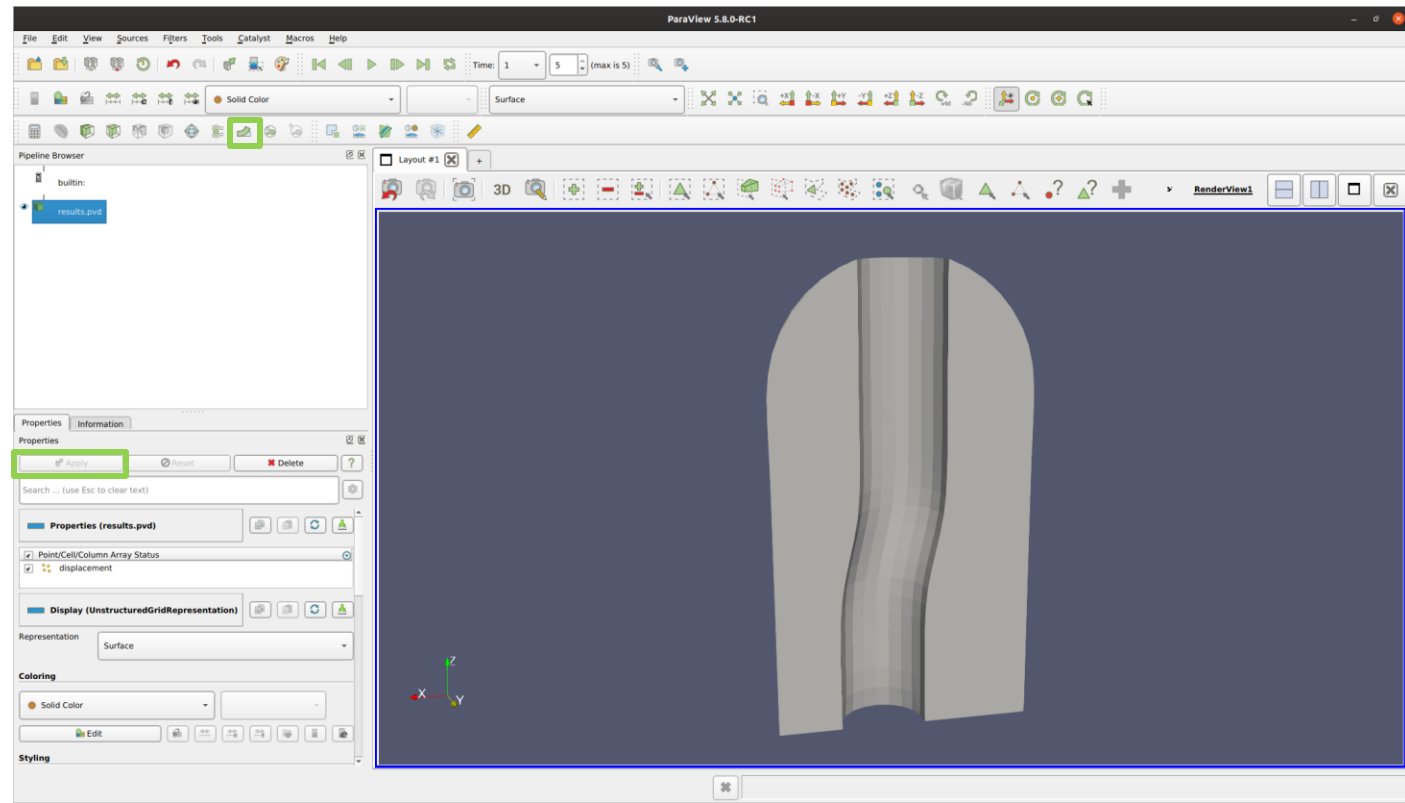
Center Axes Visibility

Orientation Axes

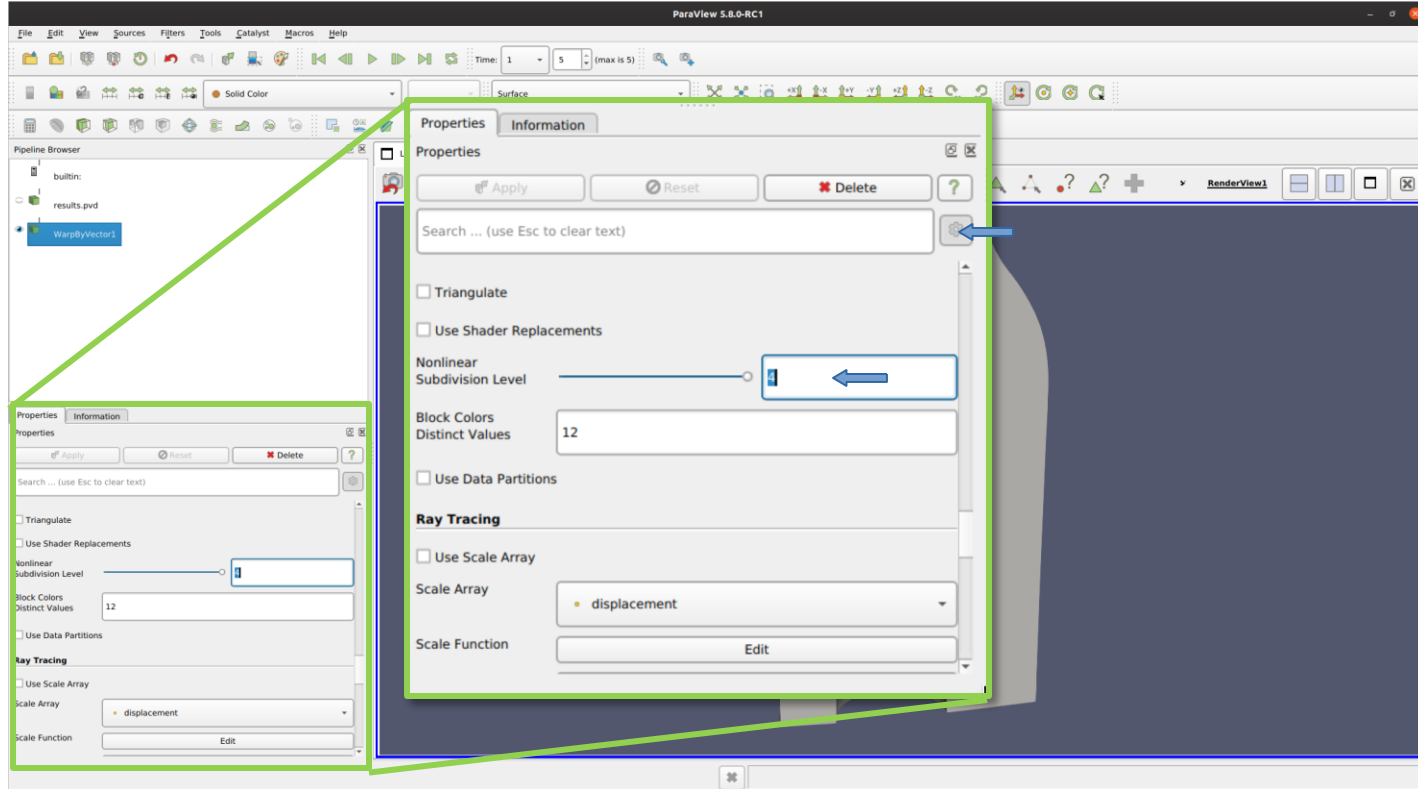
- Orientation Axes Visibility



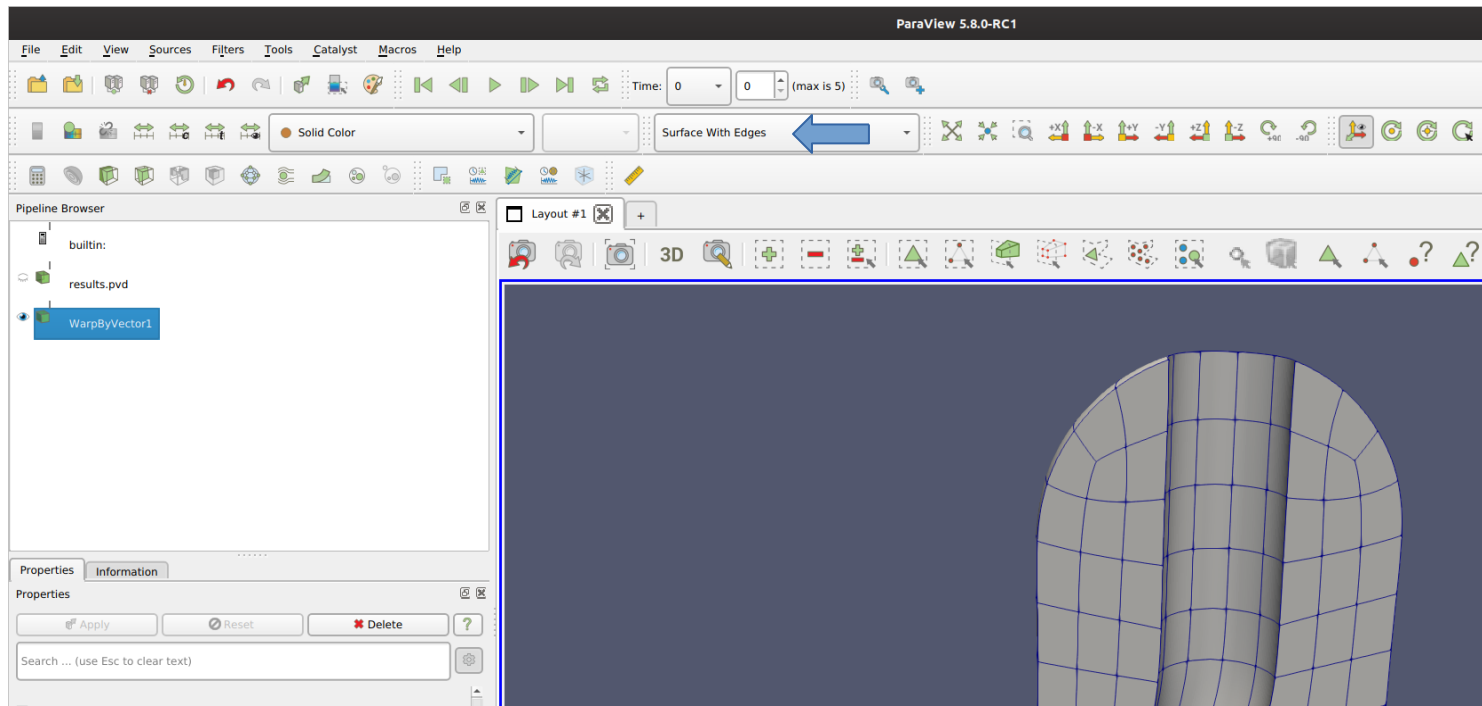
Warp By Vector



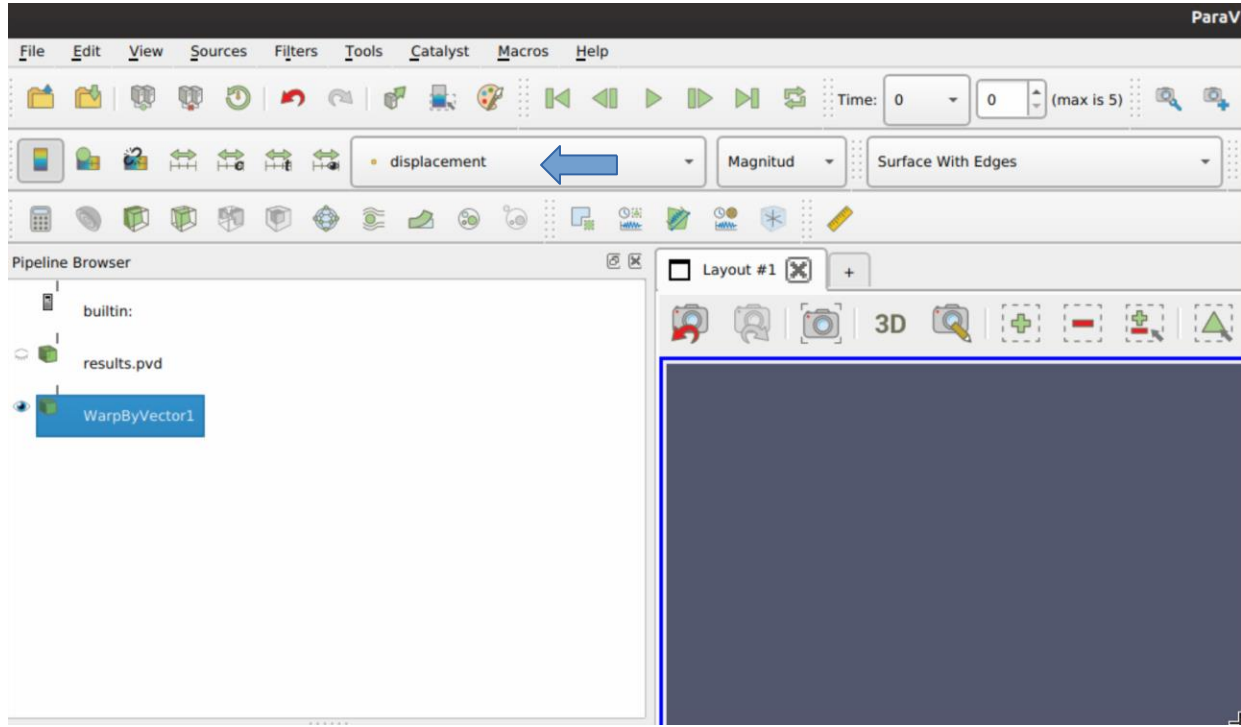
View Smooth Spline



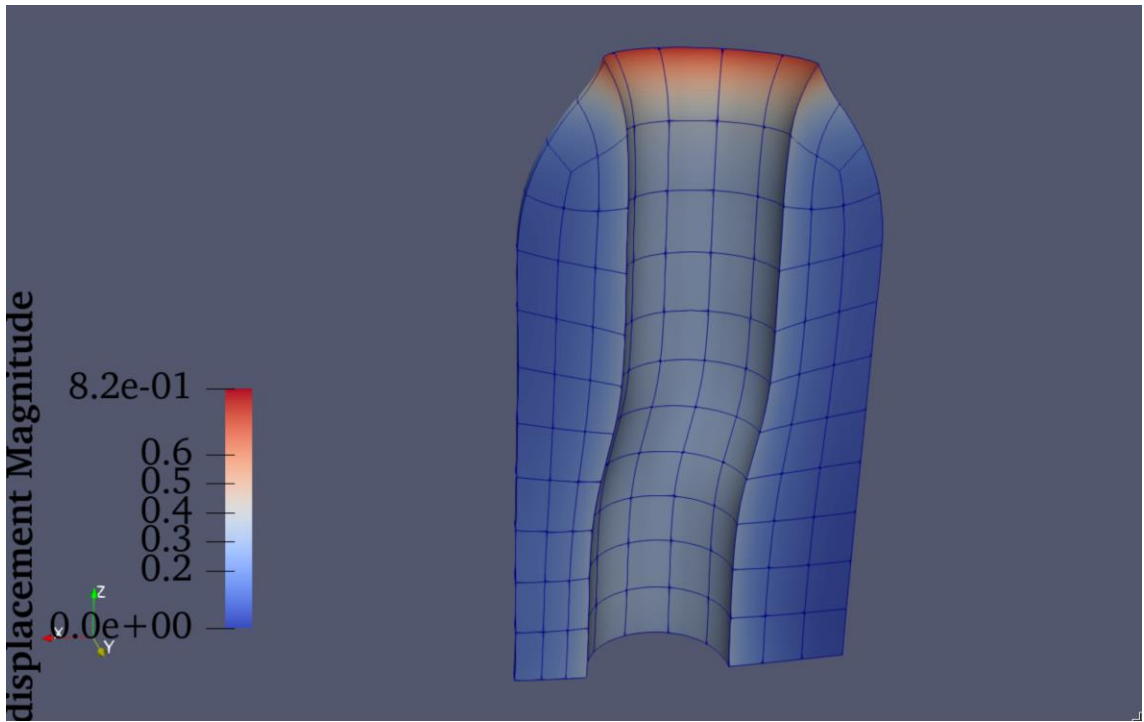
View Mesh



View Displacements

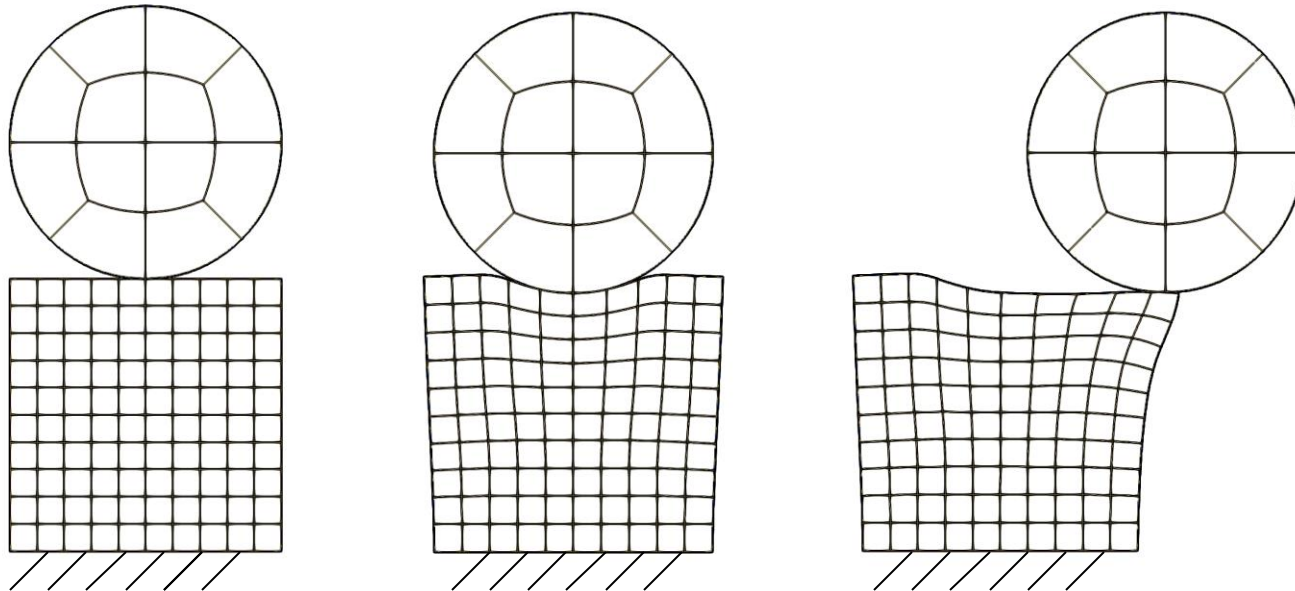


Play Displacements



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

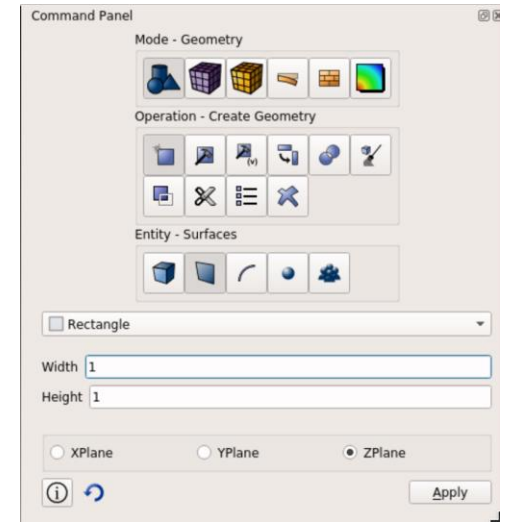
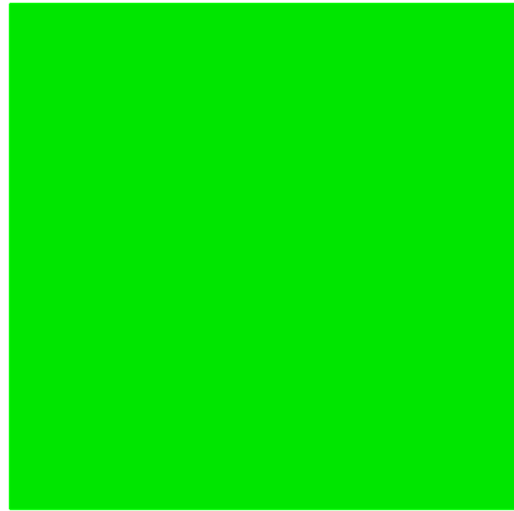
The problem will be modeled as a 2D solid quasi-static simulation

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

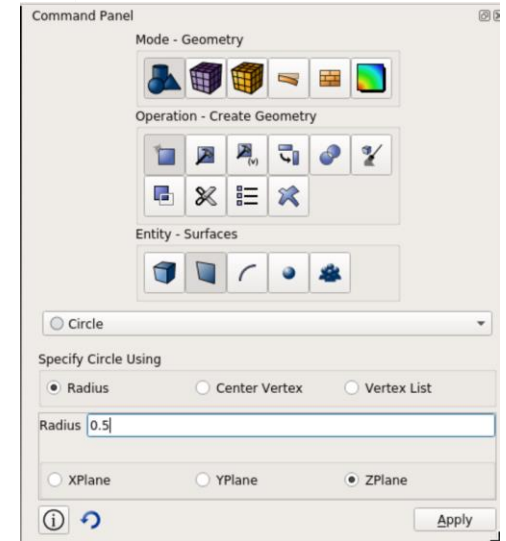
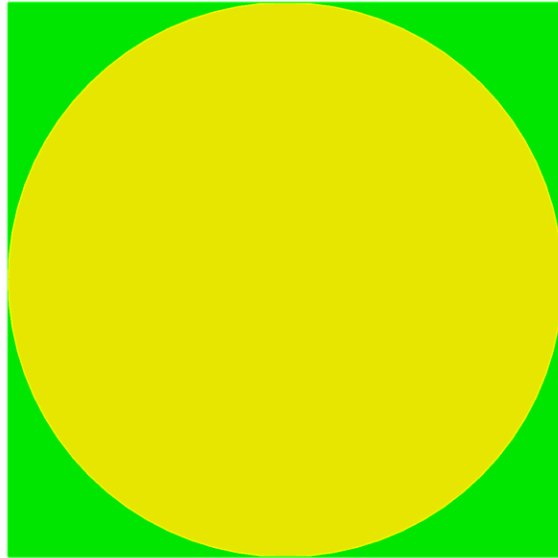
Create block



- or -

Create surface rectangle width 1 height 1 zplane

Create disc

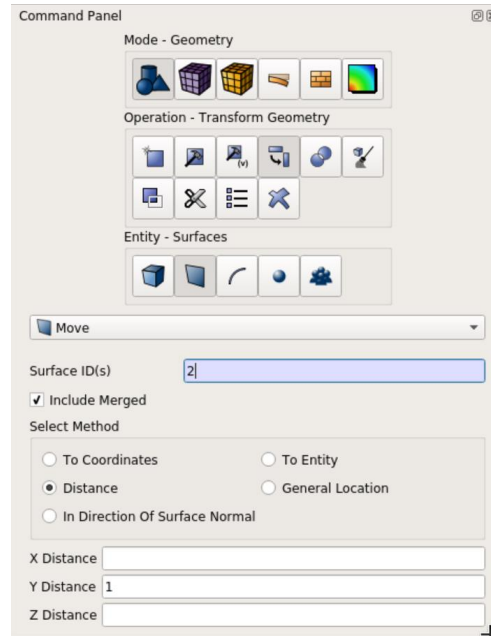
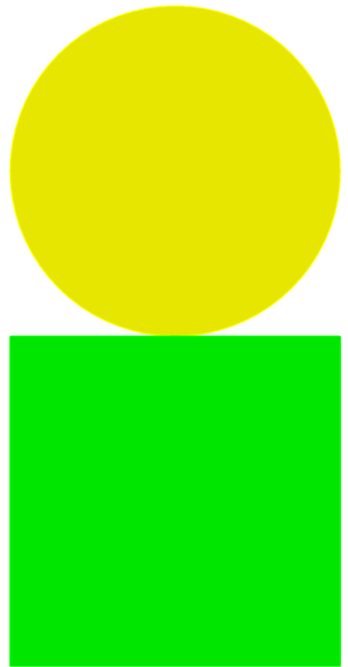


- or -

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

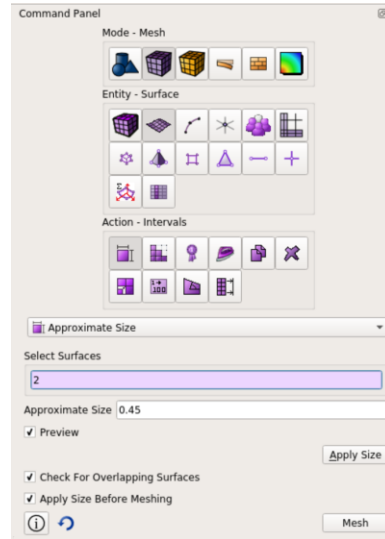
Move surface 2 y 1

Mesh disc



- or -

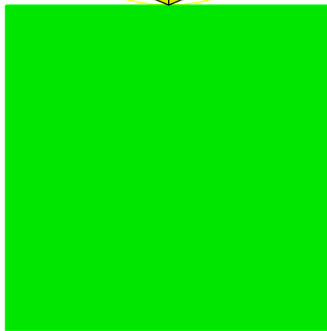
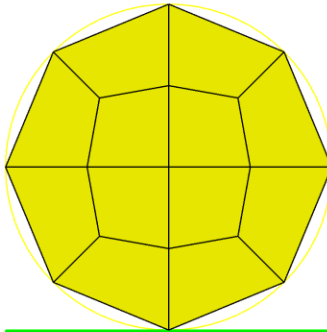
Surface 2 scheme circle



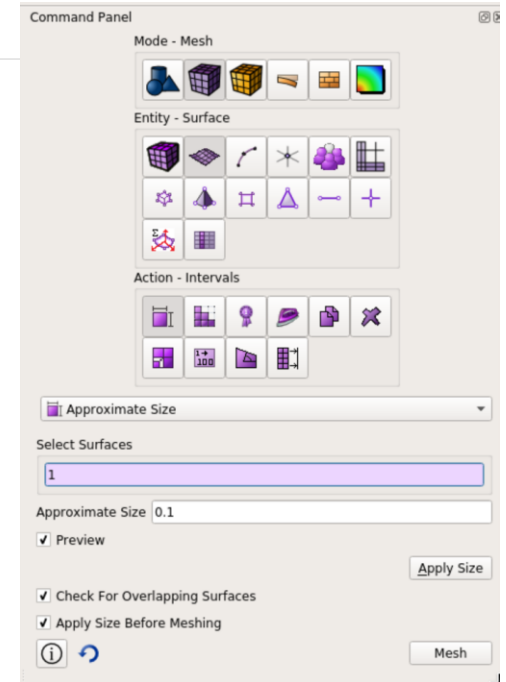
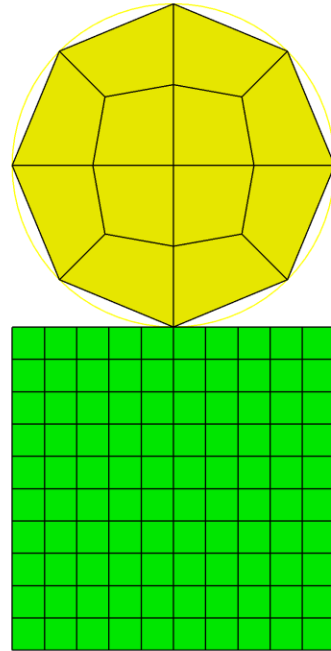
- or -

Surface 2 size 0.45
Mesh surface 2

Mesh disc

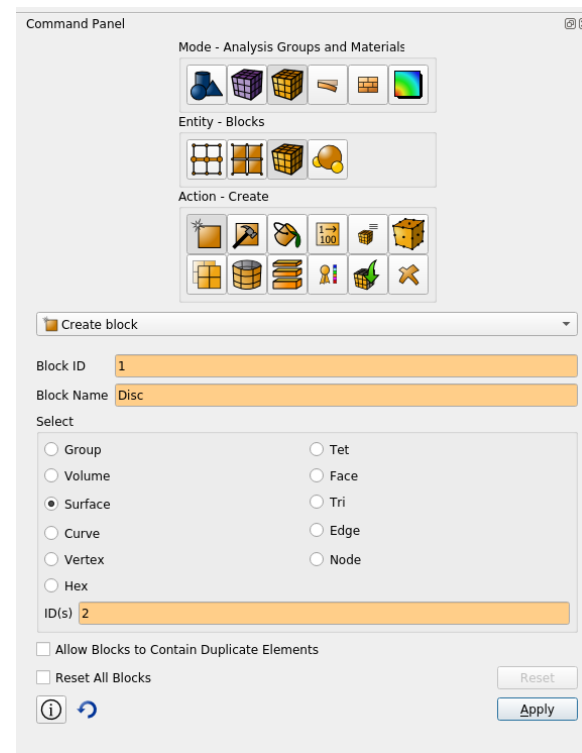
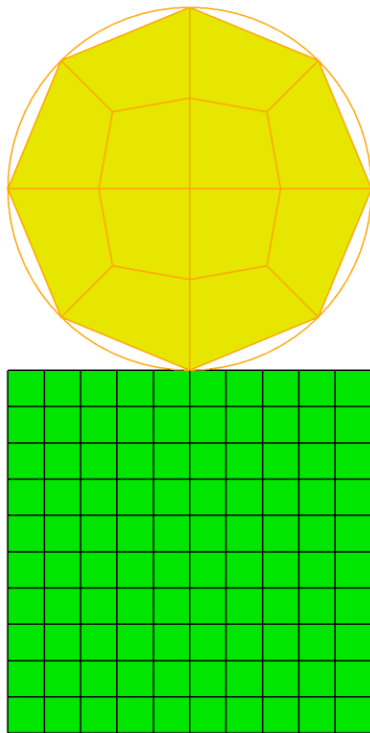


Mesh block



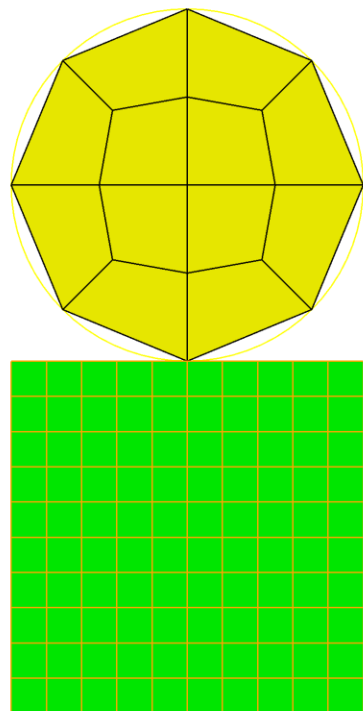
- Or -
Surface 1 size 0.1
Mesh surface 1

Create block sets - Disc



- Or -
Block 1 add surface 2
Block 1 name "Disc"

Create block sets - Block



Command Panel

Mode - Analysis Groups and Materials

Entity - Blocks

Action - Create

Create block

Block ID

Block Name

Select

- Group
- Volume
- Surface
- Curve
- Vertex
- Hex
- Tet
- Face
- Tri
- Edge
- Node

ID(s)

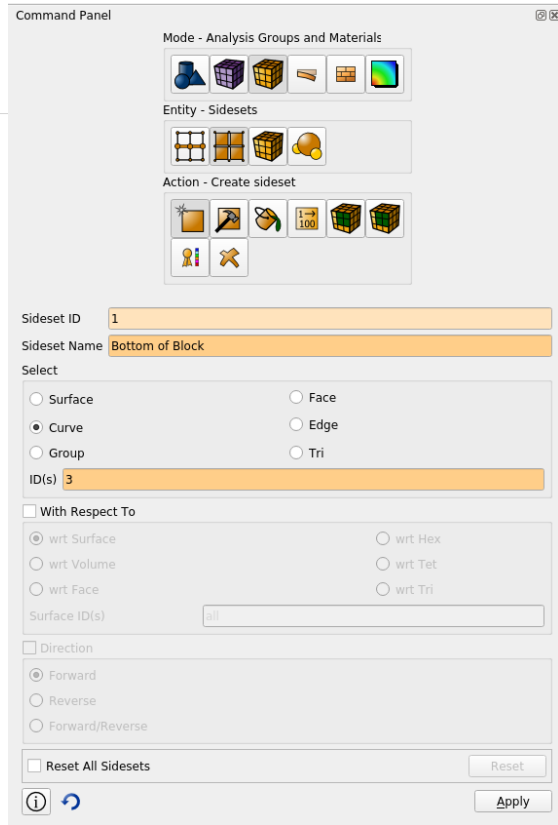
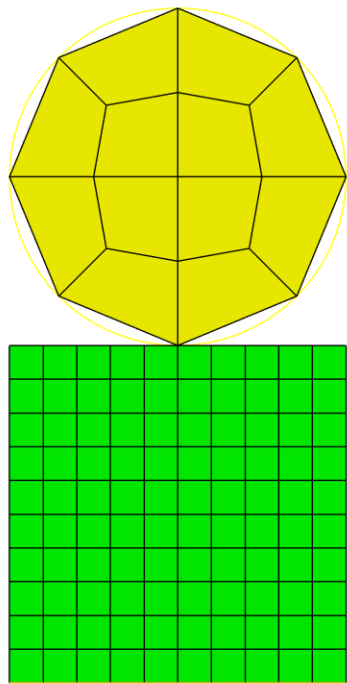
Allow Blocks to Contain Duplicate Elements

Reset All Blocks

- Or -

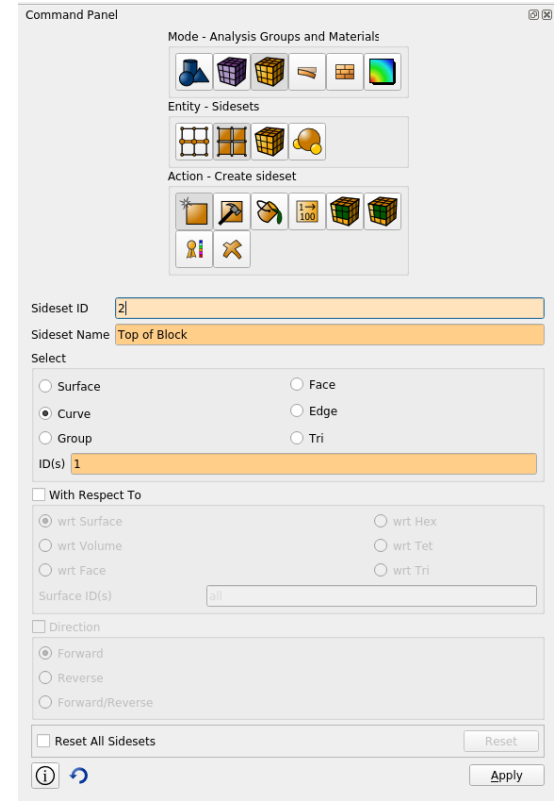
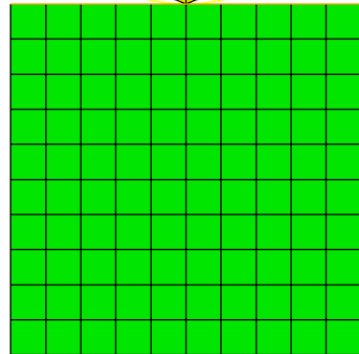
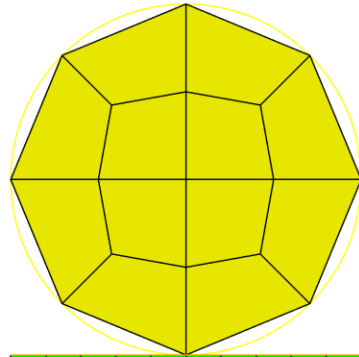
Block 2 add surface 1
Block 2 name "Block"

Create side sets bottom of block



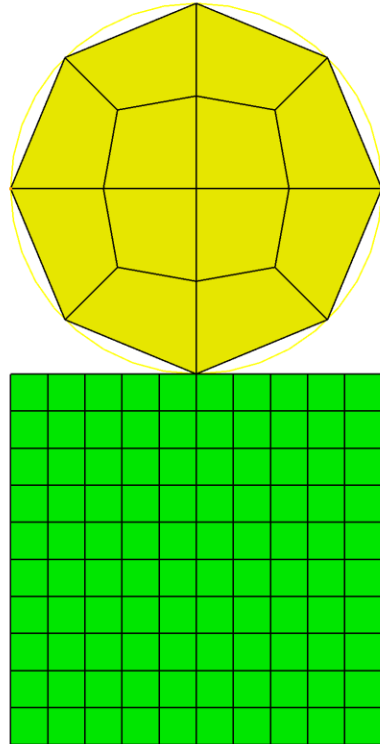
- Or -
Sideset 1 add curve 3
Sideset 1 name "Bottom of Block"

Create side sets top of block



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

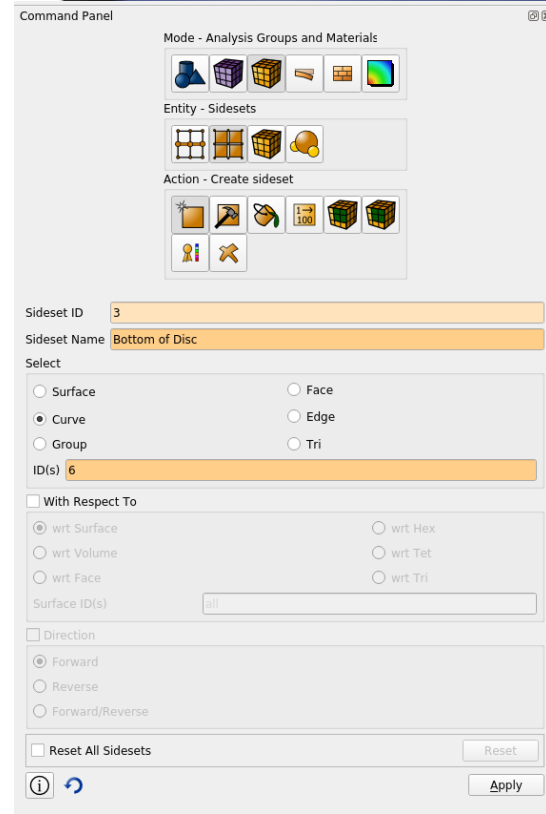
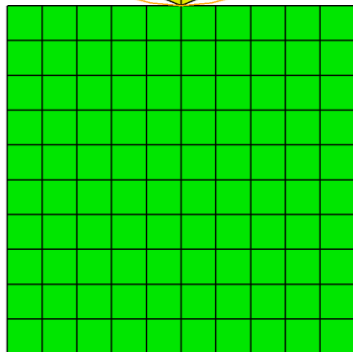
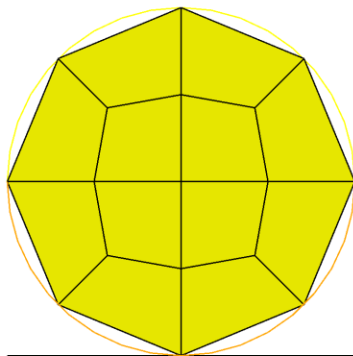
Split disc curve



- Or -

Partition create curve 5 node 1 5

Create side sets bottom of disc

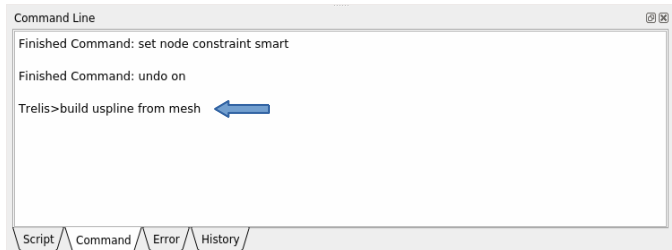


- Or -

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

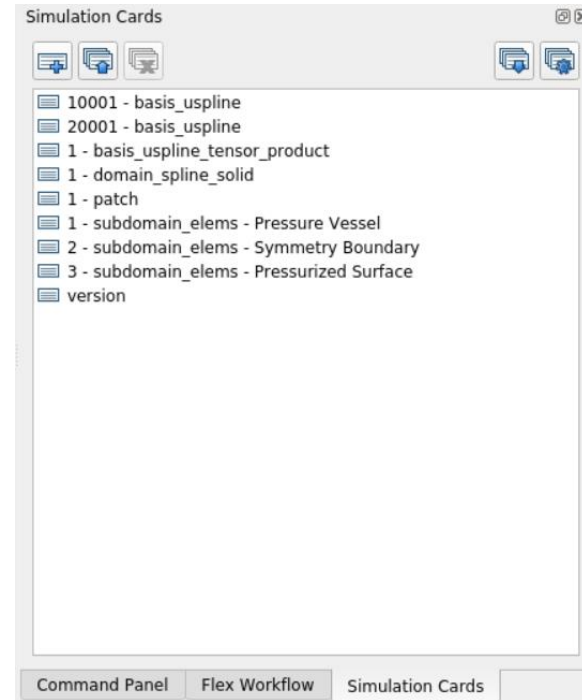
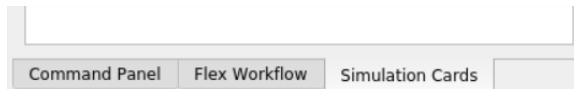
Generate and view simulation geometry cards

The simulation cards tab should now look like this:



Build is finished when Trelis> prompt returns.

Select "Simulation Cards" tab in bottom left

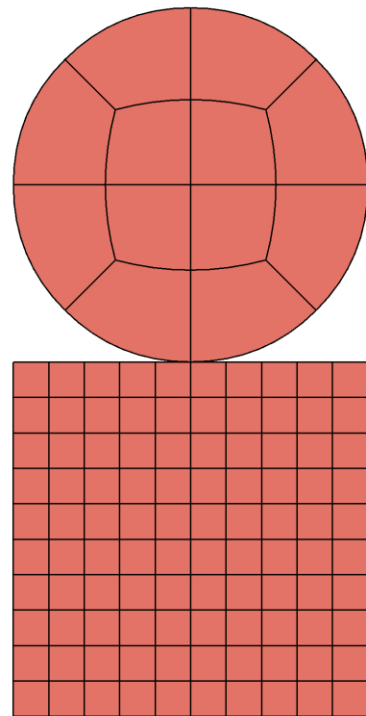


Build U-spline

```
Command Line  
Finished Command: set node constraint smart  
Finished Command: undo on  
Trelis>build uspline from mesh draw| ←
```

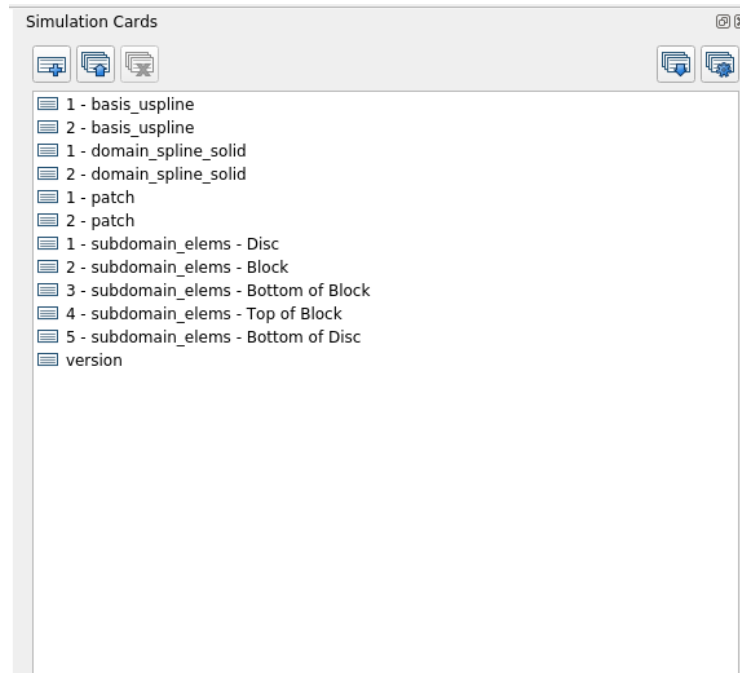
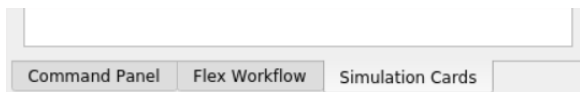
Script Command Error History

Build is finished when Trelis> prompt returns.

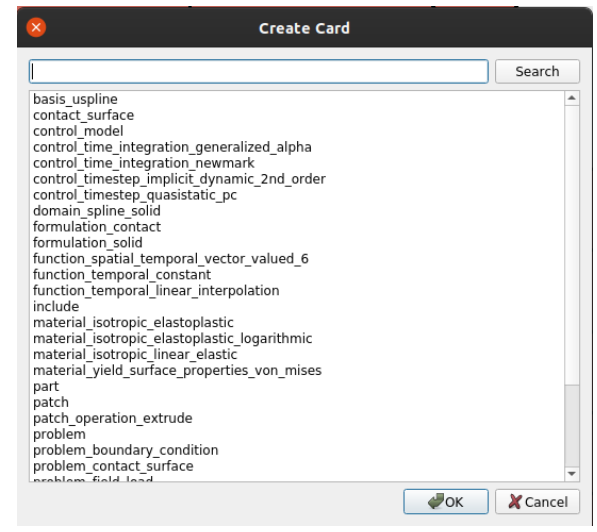
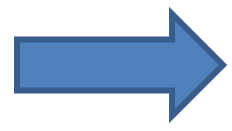
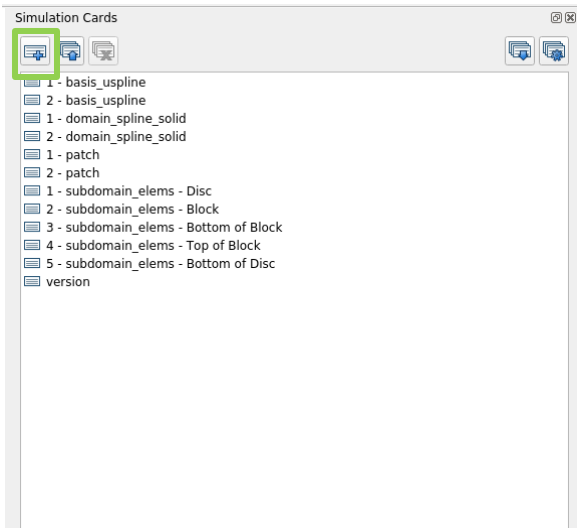


View simulation cards

Select "Simulation Cards" tab in bottom right



Creating Cards



Material definition

material_isotropic_elastoplastic_logarithmic

desc Material for both parts ←

material_id 1

E 1000 ←

E_time_dependent_function_temporal_id

E_temperature_dependent_function_temporal_id

nu 0.3 ←

rho 0 ←

material_yield_surface_properties_id 1 - Material for both parts ←

material_yield_surface_properties_von_mises

desc Material for both parts ←

material_yield_surface_properties_id 1

K 0

yield_stress 2 ←

saturation_stress

saturation_exponent 0

Element Formulation

formulation_solid	
desc	Solid formulation for both parts ←
formulation_id	1
formulation_type	solid_2d ← ▾
quadrature	QP1 ▾
material_id	1 - Material for both parts ← ▾

Parts

part

desc

part_id

formulation_id

subdomain_ids

temperature_id

part

desc

part_id

formulation_id

subdomain_ids

temperature_id

Control Timestep

control_timestep_quasistatic_pc	
desc	<input type="text"/>
control_timestep_id	<input type="text" value="1"/>
max_corrector_step_n	<input type="text" value="10"/> ←
newton_tol_abs	<input type="text" value="1e-32"/>
newton_tol_rel	<input type="text" value="1e-06"/>
delta_tol_abs	<input type="text" value="1e-32"/>
delta_tol_rel	<input type="text" value="0.001"/>
<input checked="" type="checkbox"/> line_search ←	
line_search_tol	<input type="text" value="0.5"/>
line_search_max_iterations	<input type="text" value="10"/>



Problem and Control Model

problem

desc

problem_id

part_ids

1 - Disc

2 - Block

control_timestep_id

coupled_problems

control_linear_solver

options_from_command_line

solver_type

control_model

desc

control_time

initial_time_step

termination_time

adaptive_timestep

iteration_optimal

iteration_window

growth_factor

reduction_factor

delta_t_min

delta_t_max

control_problem

enable_parent_basis

enable_output

enable_output_restart

output_restart_file_name_prefix

output_restart_delta_t

output_restart_delta_time_step

output_restart_based_on_time_step

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

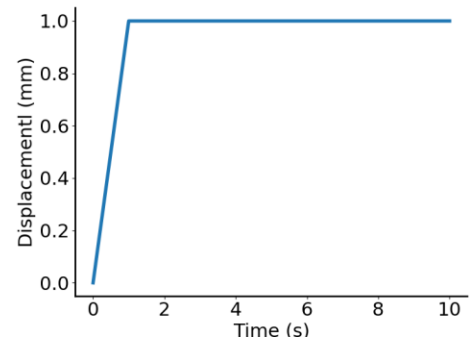
RX: 0

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 + x

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

subdomain_nodal_dva

desc: Right displacement of Disc

subdomain_nodal_value_id: 2

subdomain_id: 1 - Disc

dva_type: DISPLACEMENT

UX: 0.075

UY: 0

UZ: 0

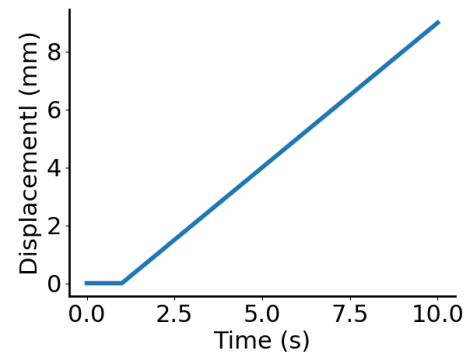
RX: 0

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 ←
<input checked="" type="checkbox"/> UX	0 ←
<input checked="" type="checkbox"/> UY	0 ←
<input type="checkbox"/> UZ	0
<input type="checkbox"/> RX	0
<input type="checkbox"/> RY	0
<input type="checkbox"/> RZ	0
<input checked="" type="radio"/> function_temporal_id	3 - Always 1 ←
<input type="radio"/> function_spatial_temporal_id	

Problem boundary condition

problem_boundary_condition

desc

problem_id

subdomain_nodal_value_ids + X

1 - Downward displacement of disc	←
2 - Right displacment of Disc	←
3 - Clamp bottom of block	←

Set up contact

formulation_contact

desc Disc to block ←

formulation_id 2

formulation_type contact_gpts_2d ←

penalty 100000 ←

use_soft_contact

gap_tolerance 1e-06

nearest_point_tolerance 1e-12 ←

nearest_point_max_iteration_n 10

master_seed_points_n 10

search_sphere_tolerance 0.1

search_cone_tolerance 0 ←

master_seeds QP1

slave_quadrature QP1

contact_surface

desc Disc to block ←

contact_surface_id 1

formulation_id 2 - Disc to block ←

slave_subdomain_id 4 - Top of Block ←

master_subdomain_id 5 - Bottom of Disc ←

problem_contact_surface

desc Deep rolling - disc to block ←

problem_id 1 - Deep rolling contact ←

contact_surface_ids

1 - Disc to block ←

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

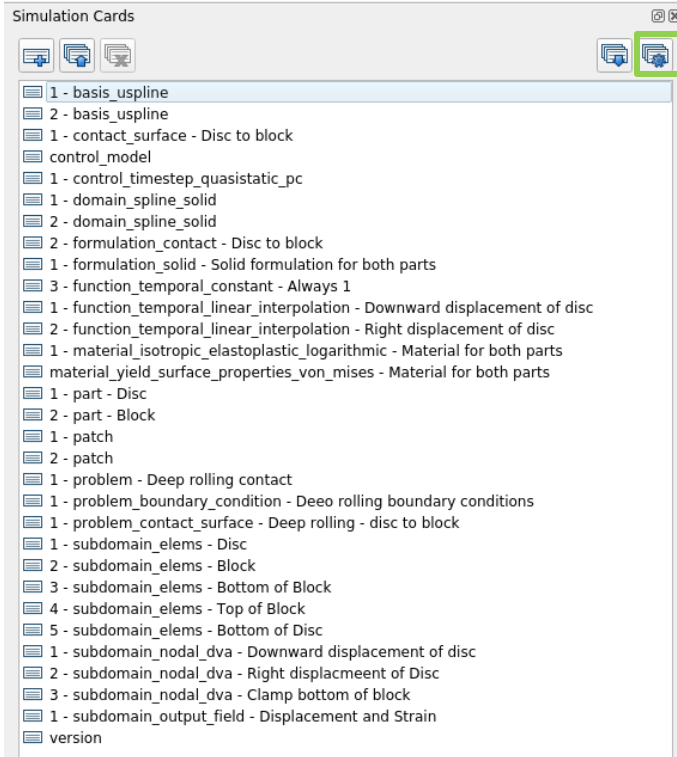
include_elem_outlines

solution_type: current

bezier_projection_relative_continuity:

Run Simulation

Simulation Cards



- 1 - basis_uspline
- 2 - basis_uspline
- 1 - contact_surface - Disc to block
- control_model
- 1 - control_timestep_quasistatic_pc
- 1 - domain_spline_solid
- 2 - domain_spline_solid
- 2 - formulation_contact - Disc to block
- 1 - formulation_solid - Solid formulation for both parts
- 3 - function_temporal_constant - Always 1
- 1 - function_temporal_linear_interpolation - Downward displacement of disc
- 2 - function_temporal_linear_interpolation - Right displacement of disc
- 1 - material_isotropic_elastoplastic_logarithmic - Material for both parts
- material_yield_surface_properties_von_mises - Material for both parts
- 1 - part - Disc
- 2 - part - Block
- 1 - patch
- 2 - patch
- 1 - problem - Deep rolling contact
- 1 - problem_boundary_condition - Deeo rolling boundary conditions
- 1 - problem_contact_surface - Deep rolling - disc to block
- 1 - subdomain_elems - Disc
- 2 - subdomain_elems - Block
- 3 - subdomain_elems - Bottom of Block
- 4 - subdomain_elems - Top of Block
- 5 - subdomain_elems - Bottom of Disc
- 1 - subdomain_nodal_dva - Downward displacement of disc
- 2 - subdomain_nodal_dva - Right displacment of Disc
- 3 - subdomain_nodal_dva - Clamp bottom of block
- 1 - subdomain_output_field - Displacement and Strain
- version

Save cards and run analysis

Look in: /home/mitch/WIP/short...g/DeepRollingComplete

Computer	Name	Size	Typ
WIP			
mitch			

File name: DeepRollingComplete

Files of type: *.json

Run Simulation

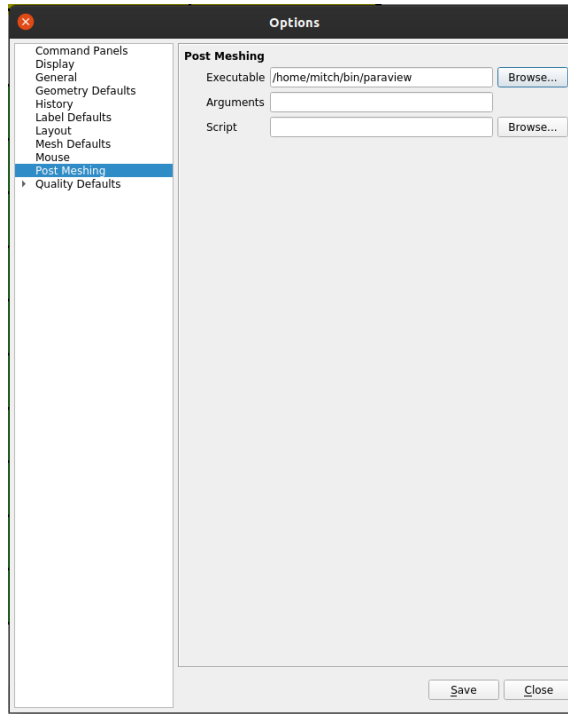
```
Command Line
-----
-----> Computing external force...
-----> Computing internal force...
-----> Checking residual for convergence
-----> Abs Res. norm: 0.0349368 tol: 1e-32
-----> Rel Res. norm: 0.0237485 tol: 1e-06
-----> Abs Delta norm: 0.00125713 tol: 1e-32
-----> Rel Delta norm: 1.30572 tol: 0.001
Line search iter:
g0: 6.53857e-06
p0: -4.97345e-06
-----> Begin line search...
pa * g0: -3.25192e-11
sa: 1
sb: 0
g0: 6.53857e-06
p0: -4.97345e-06
j0: 6.53857e-06
-----> Computing external force...
-----> Computing internal force...
-----> Checking residual for convergence
-----> Abs Res. norm: 0.0158858 tol: 1e-32
-----> Rel Res. norm: 0.0158858 tol: 1e-06
-----> Abs Delta norm: 0.00125713 tol: 1e-32
-----> Rel Delta norm: 1.30572 tol: 0.001
sa: 0.567978
sb: 0
g0: 6.53857e-06
p0: 8.76681e-07
pa: 8.76681e-07
j0: -4.97345e-06
-----> Assembling the stiffness matrix...
-----> Solving the linear system...
```

View Results

```
-----> Computing external force...
-----> Computing internal force...
-----> Checking residual for convergence
----->>> Abs Res. norm: 8.82657e-13 tol: 1e-32
----->>> Rel Res. norm: 6.9234e-11 tol: 1e-06
----->>> Abs Delta norm: 1.12011e-08 tol: 1e-32
----->>> Rel Delta norm: 0.000151879 tol: 0.001
----->>> CONVERGED in 3 steps
-----> Writing output...
Writing to file: results_ts000900.vtu
-----
----->>> ABAQUS/POST PROCESSING TIME REPORT
-----
Total elapsed time (secs): 330.107
Output (secs): 2.13609 (0.633268%)
Input (secs): 0 (0%)
Report for problem 1
Total problem time (secs): 333.533 (99.2076%)
Time integrator (secs): 333.488 (99.9896%)
Correction iterations (secs): 151758 (45.4825 %)
External F assembly (secs): 1.08201 (0.317348 %)
Internal F assembly (secs): 28.2210 (8.45187 %)
Constraint assembly (secs): 64.342 (19.2972 %)
Total assembly (secs): 105.537 (31.7964 %)
Solver (secs): 66.9272 (20.066 %)
ParentBasis: timer 0: 0
PatchBasis: timer 0: 2.3097
ParentBasis: timer 0: 0
PatchBasis: timer 0: 0.486223
Solver process exited normally with exit code 0
```

Cancel job Post Process Close

Set Post Process Executable



Tools -> Options -> Post Meshing

View Results

Pipeline Browser

- built:
- results.pvd

Properties Information

Apply Search Delete ?

Search (Use Esc to clear text)

Properties (results.pvd)

- PointCloudColumn Array Status
- displacement
- top

Display

View (Render View)

Axes Grid Edit

Center Axes Visibility

Orientation Axes

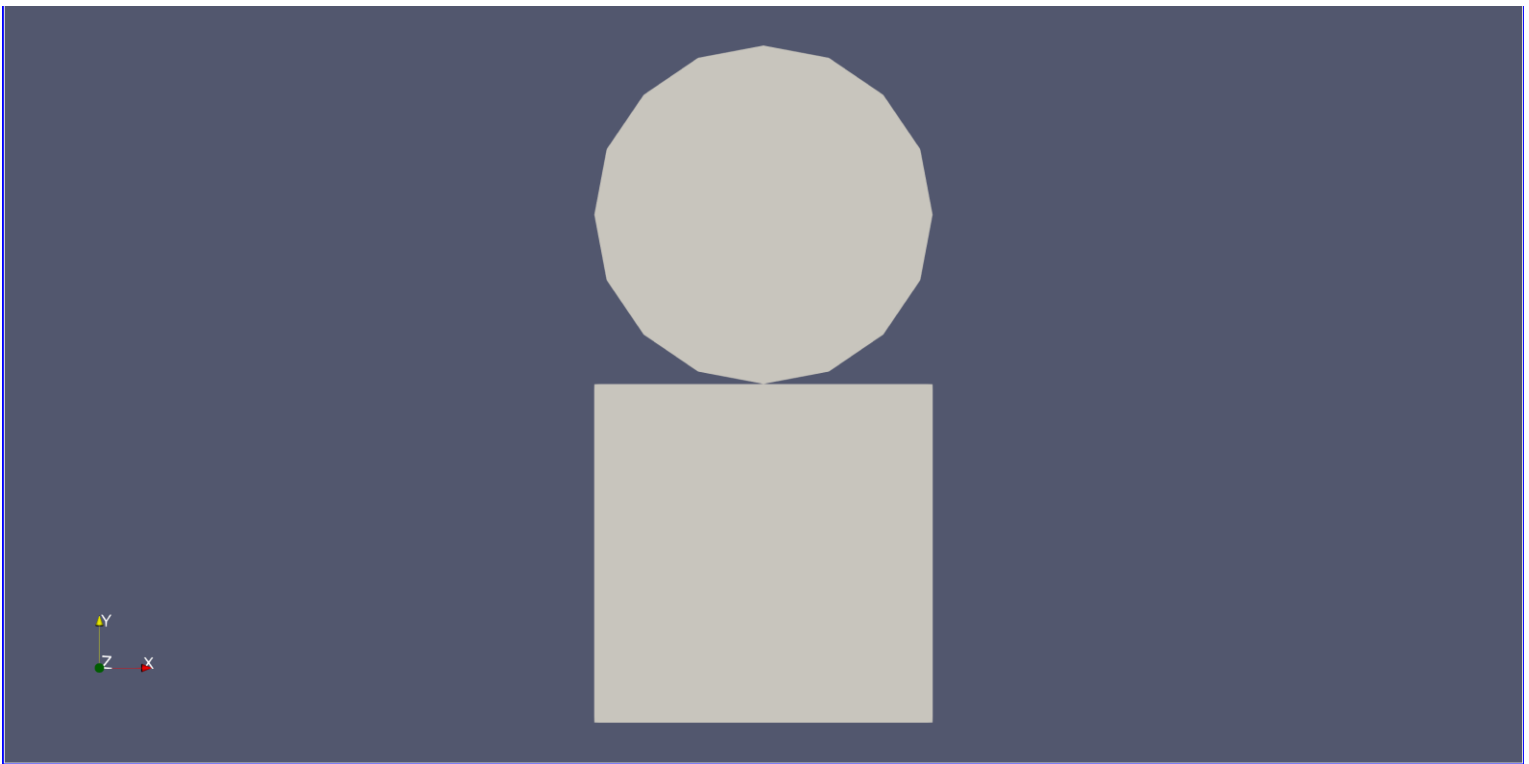
- Orientation Axes Visibility

Hidden Line Removal

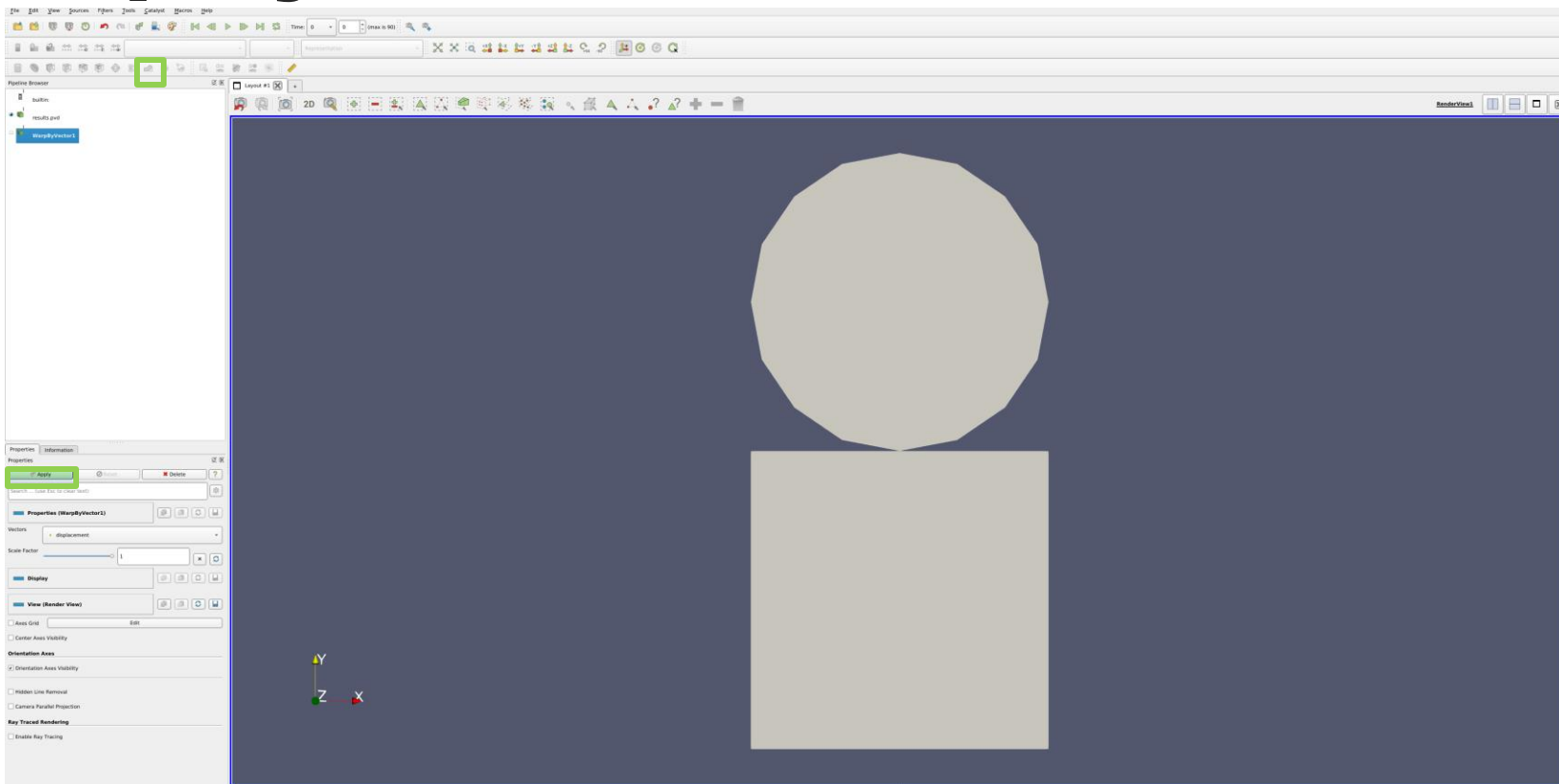
- Camera Parallel Projection

Ray Traced Rendering

- Enable Ray Tracing

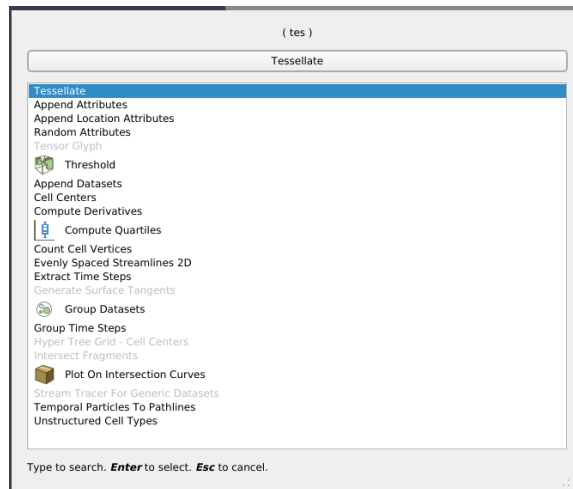


Warp By Vector



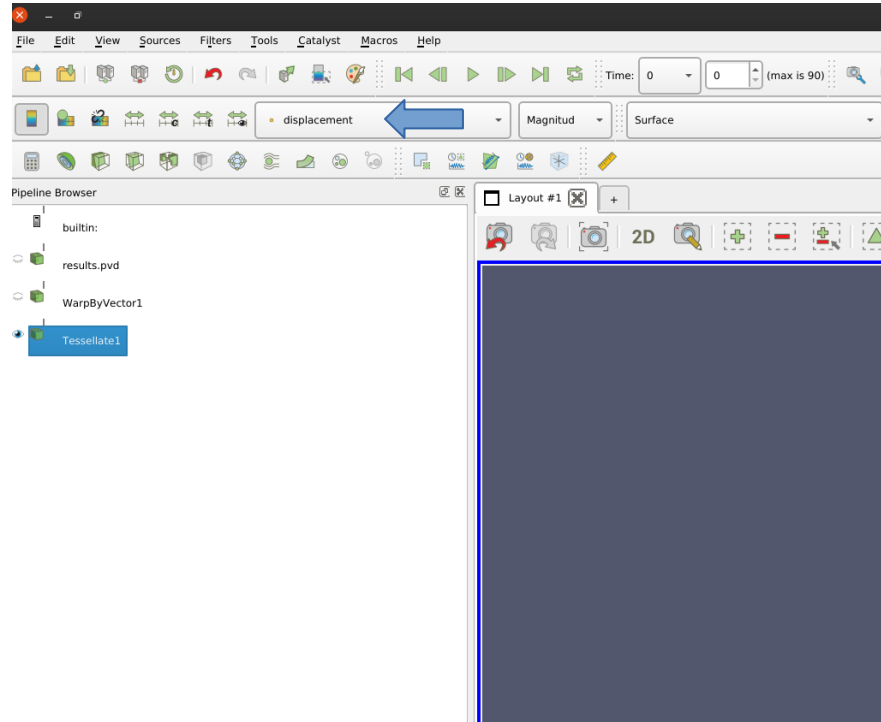


Tessellate

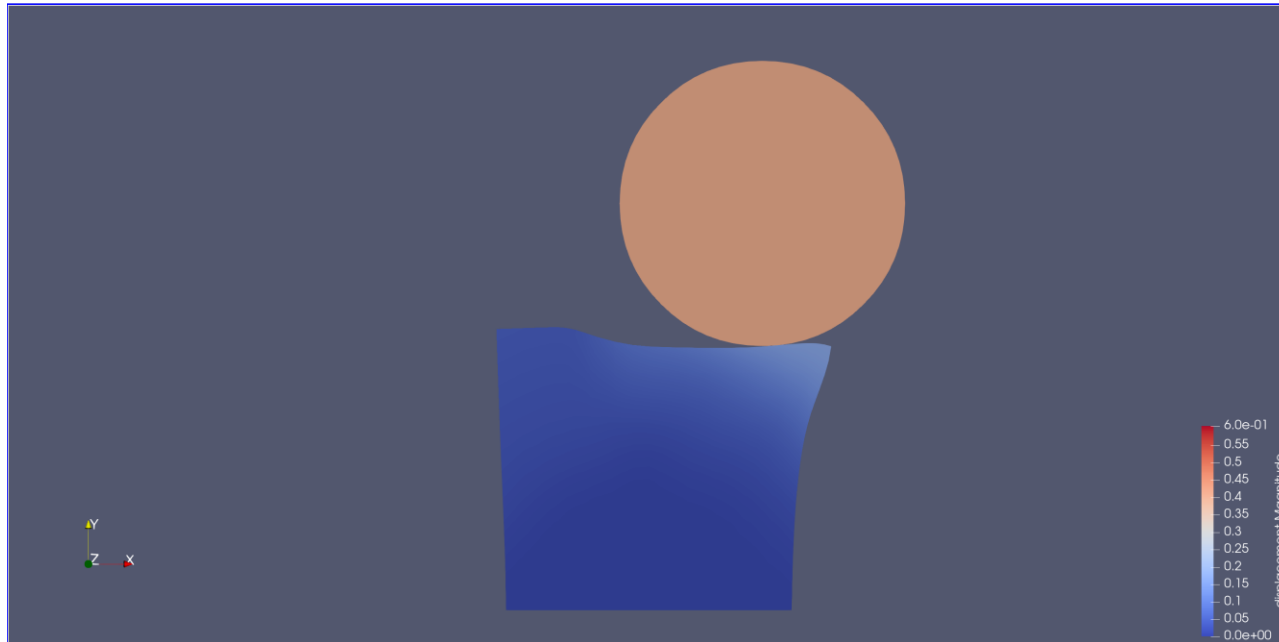


Filters -> Search -> Tesselate
Then click apply

View Displacements

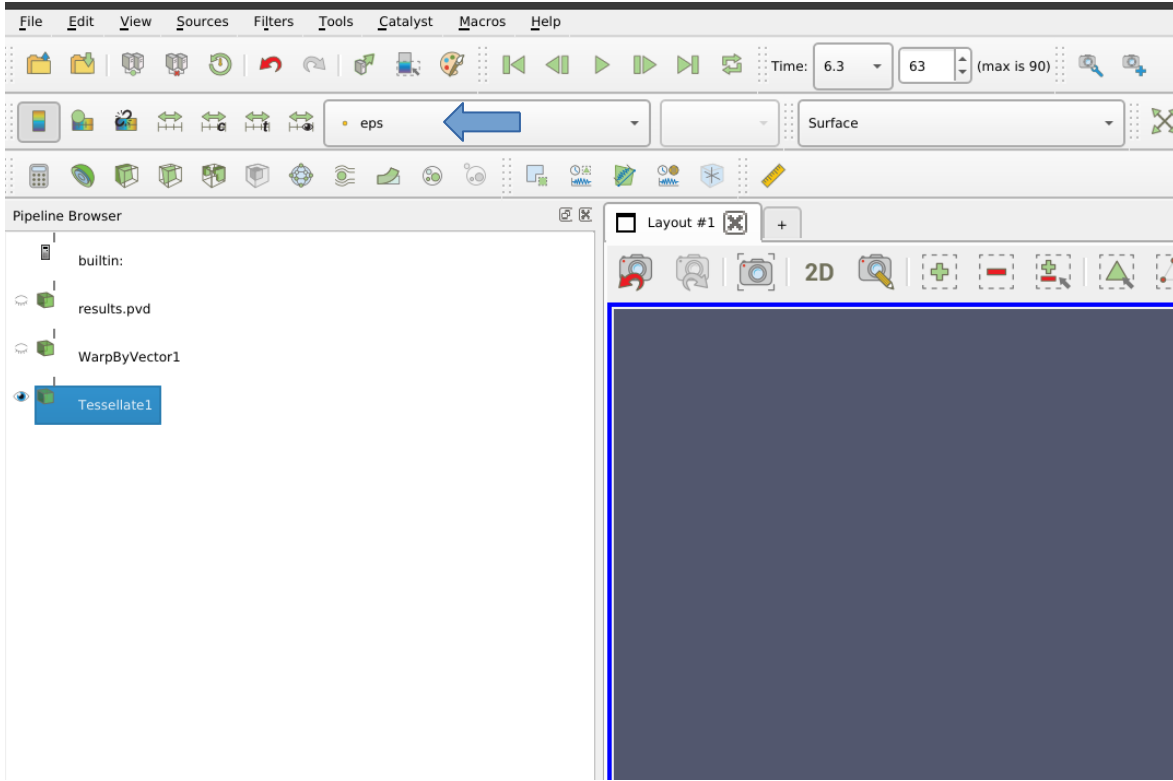


Play Displacements

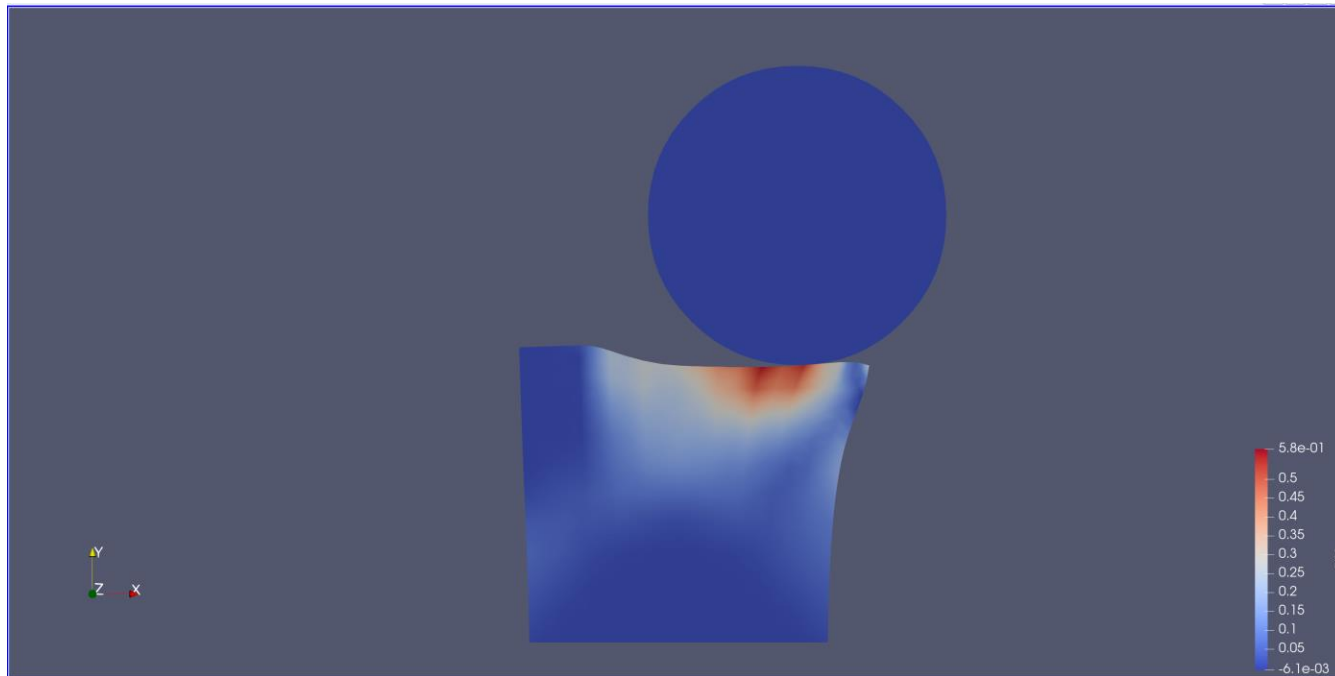


You may need to adjust the color scaling

View eps

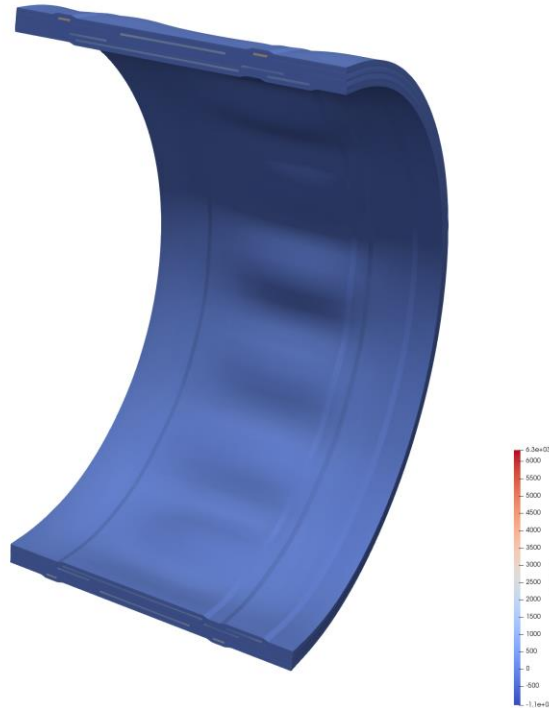


Play eps



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.

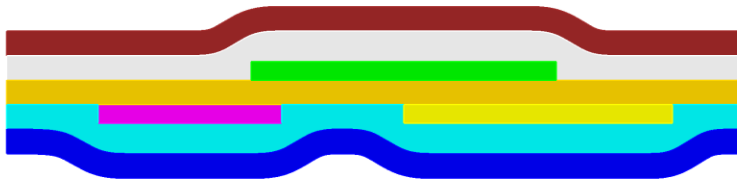
A custom workflow has been created to simulate this problem. The first step is to create a U-spline of the cable cross section. The cross section is then extruded to form a 3D solid and the boundary conditions are applied. The extrusion and application of boundary conditions are automated using a Python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

Learning Objectives

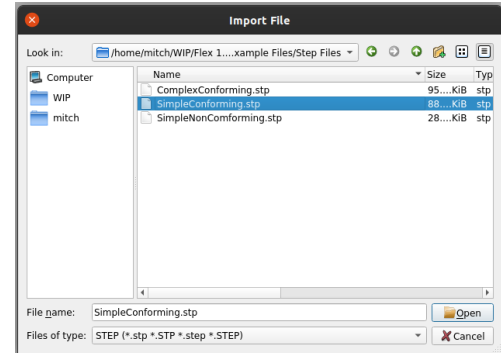
Tutorial participants will learn:

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

Import geometry



File->Import

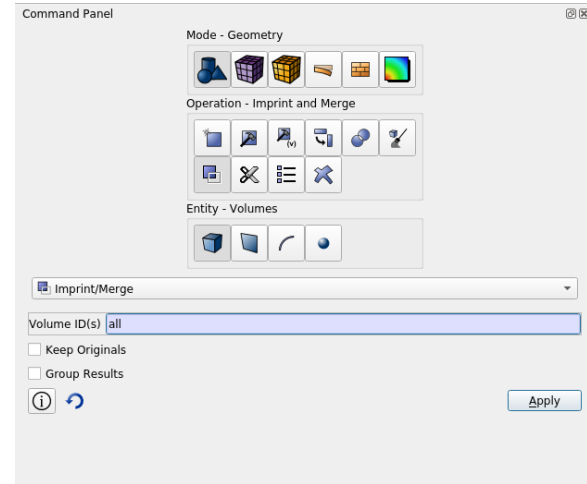
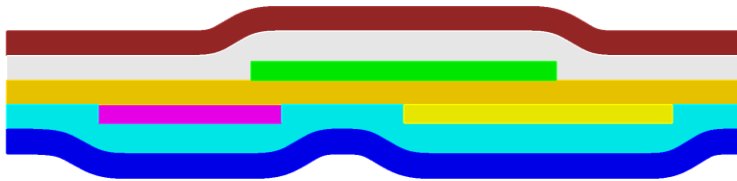


Accept import default options

- or -

Import step "<path>" heal

Imprint and merge

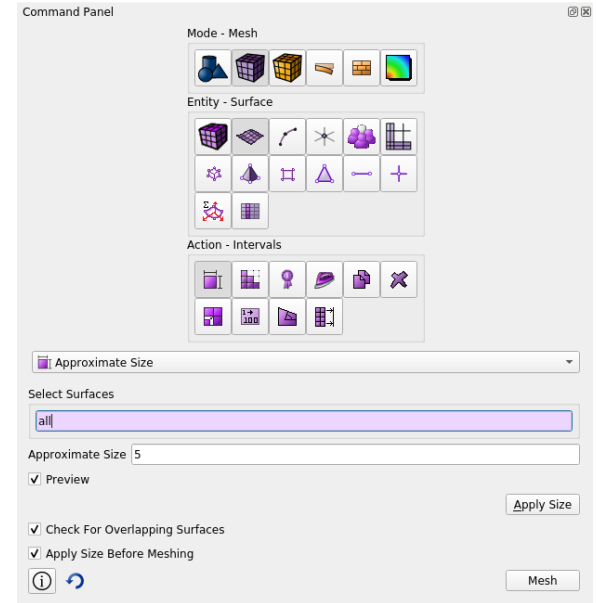
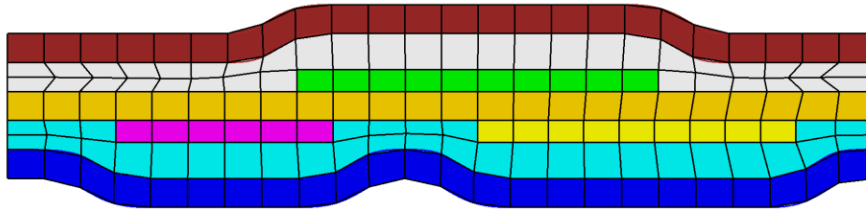


- or -

Imprint all
Merge all



Set mesh size then mesh surface

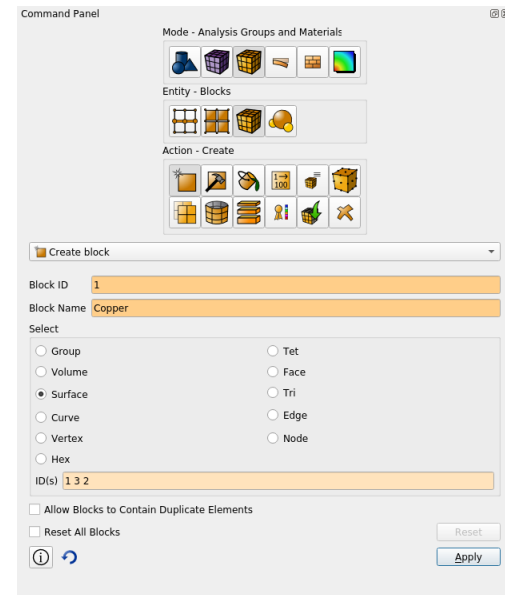
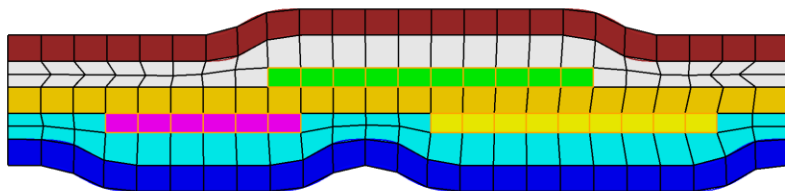


- Or -

surface all size 5
Mesh surface all



Setup Trelis Blocks - Copper

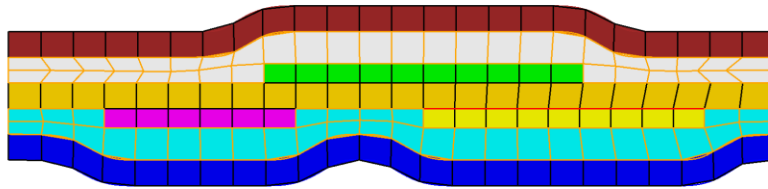


- Or -

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



Setup Trelis Blocks - Adhesive



Command Panel

Mode - Analysis Groups and Materials

Entity - Blocks

Action - Create

Create block

Block ID

Block Name

Select

Group Tet

Volume Face

Surface Tri

Curve Edge

Vertex Node

Hex

ID(s)

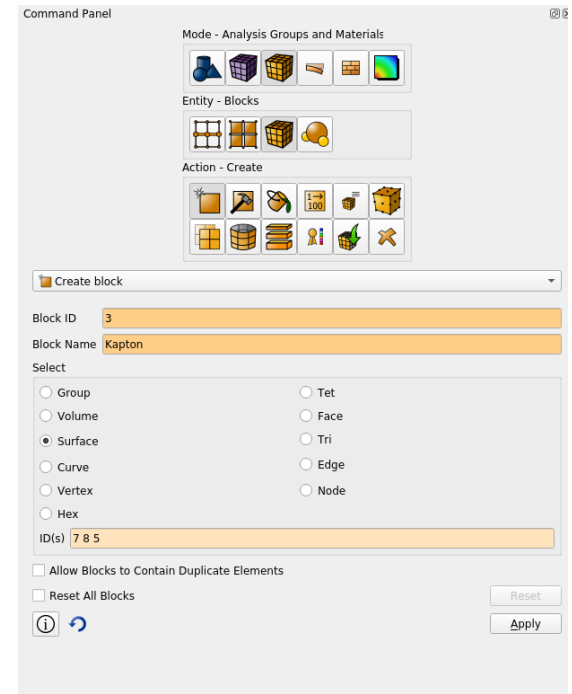
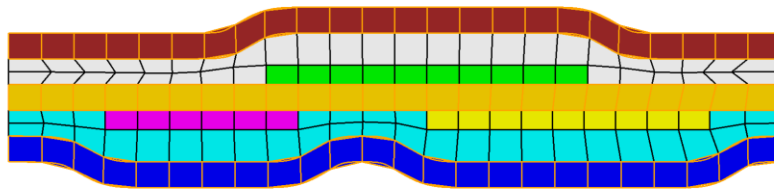
Allow Blocks to Contain Duplicate Elements

Reset All Blocks

- or -

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

Setup Trelis Blocks - Kapton

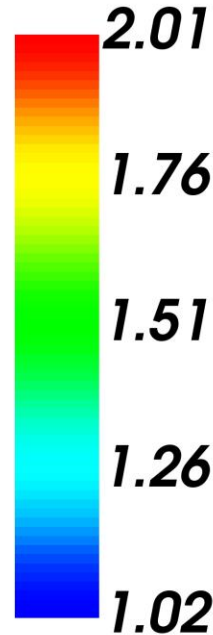


- Or -

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



Check strip quality



Flex Workflow
Steps - Mesh Strip Quality

Body Id(s)

Analyze Mesh

Worst Edges

Edge ID	Length Ratio
454	2.006648
479	2.006648
502	1.896896
486	1.816941
517	1.743831
577	1.737403
602	1.737403
500	1.664681
615	1.638608
649	1.602681

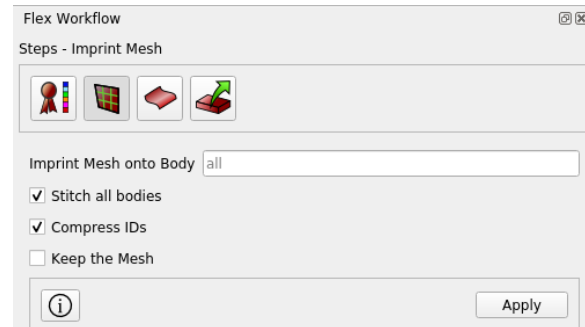
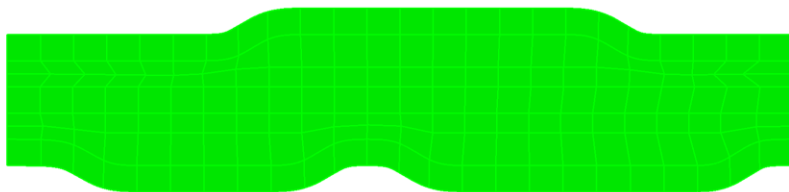
Draw Strip

- or -

Uspline strip quality body all



Imprint mesh onto geometry

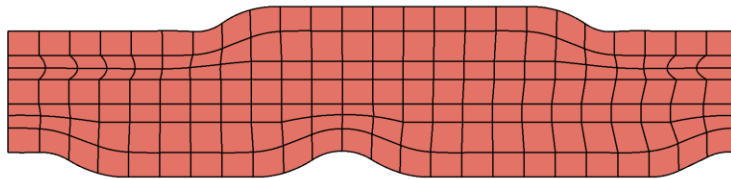


- or -

Imprint mesh onto body all
Stitch body all
compress



Build U-spline



Flex Workflow ⊞ ⊞

Steps - U-spline Commands

Commands - Build U-spline

Body Id(s)

Degree

Default Continuity

Domain Type

Convert Blocks to Subdomains

Create Subdomain Boundaries

Draw U-spline

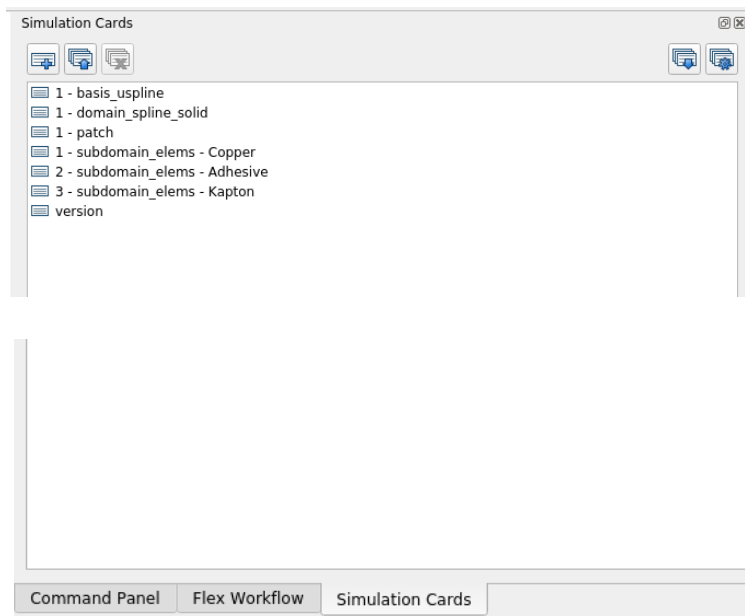
Show Continuity

ⓘ Build U-spline

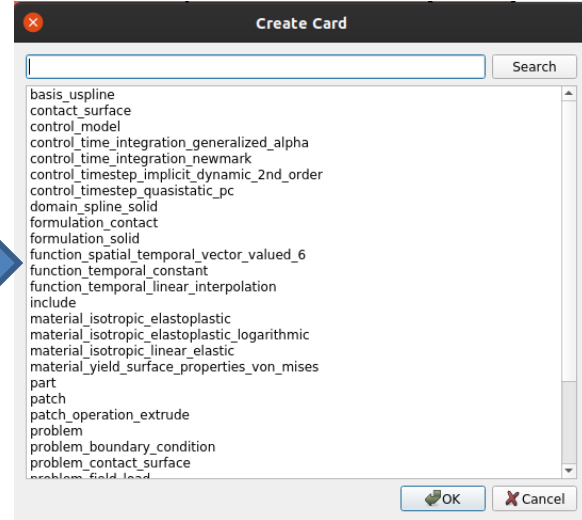
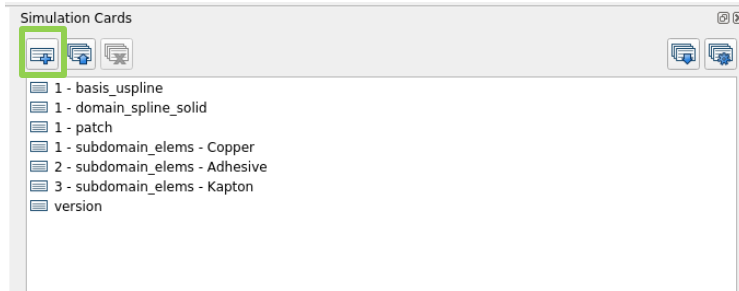
- Or -

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

View simulation cards



Create card



Add material for adhesive

function_temporal_linear_interpolation

desc Adhesive Modulus

function_temporal_id 1

birth 70

death 160

tol 1e-10

graph + x

	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

formulation_solid

desc

formulation_id

formulation_type

quadrature

material_id

part

desc

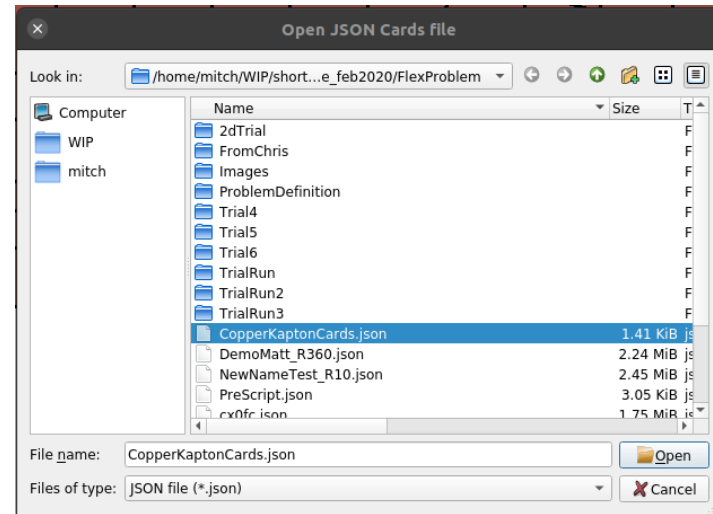
part_id

formulation_id

subdomain_ids

temperature_id

Import cards for copper and kapton



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

Associate imported parts with correct subdomain

part

desc

part_id

formulation_id

subdomain_ids

temperature_id

part

desc

part_id

formulation_id

subdomain_ids

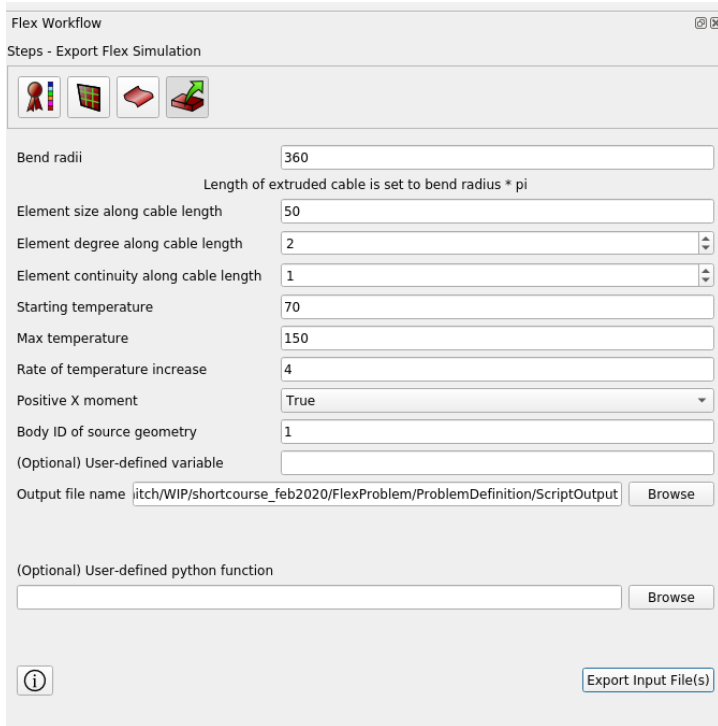
temperature_id

Add output cards

function_temporal_constant	
desc	Always 1 ←
function_temporal_id	2
value	1
birth	0
death	10000000000
tol	1e-10


subdomain_output_field	
desc	
subdomain_output_id	1
subdomain_ids	<div style="border: 1px solid gray; padding: 2px;"><p>1 - Copper ←</p><p>2 - Adhesive ←</p><p>3 - Kapton ←</p></div>
function_temporal_id	2 - Always 1 ←
field_types	<ul style="list-style-type: none"><input checked="" type="checkbox"/> displacement ←<input type="checkbox"/> velocity<input type="checkbox"/> acceleration<input type="checkbox"/> strain<input type="checkbox"/> stress<input type="checkbox"/> vm_stress<input type="checkbox"/> pressure<input type="checkbox"/> contact_pressure<input type="checkbox"/> contact_gap<input type="checkbox"/> eps<input type="checkbox"/> effective_plastic_work<input type="checkbox"/> effective_driving_energy<input type="checkbox"/> phase_field<input type="checkbox"/> phase_field_rate<input type="checkbox"/> weight
<input checked="" type="radio"/> delta_time	0.1 ←
<input type="radio"/> delta_step	
file_name_prefix	results
file_type	vtk
sample_type	BEZIER ←
<input checked="" type="checkbox"/> cache_basis_evals	
<input checked="" type="checkbox"/> include_elem_outlines	
solution_type	current
bezier_projection_relative_continuity	

Export Flex simulation file



Flex Workflow

Steps - Export Flex Simulation



Bend radii

Length of extruded cable is set to bend radius * pi

Element size along cable length

Element degree along cable length

Element continuity along cable length

Starting temperature

Max temperature

Rate of temperature increase

Positive X moment

Body ID of source geometry

(Optional) User-defined variable

Output file name

(Optional) User-defined python function

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

