On Setting up a 2D Structured ALE Model

Hao Chen, Ansys

LS-DYNA ALE has been widely used to simulating moving fluids interacting with structures. Unlike CFD, the focus is rather on the structure response under dynamic loading from fluids, than the fluids' motion. Fluids are agitated by a high pressure gradient; and then hit the structure, carrying a large momentum. The key in successfully capturing the physics lies in the fluid-structure interaction algorithm. It needs to accurately predict the peak of pressure loading during the impact, which is characterized as a momentum transfer process. This request could only be fulfilled by a transient analysis with a penalty-based coupling between fluids and structure.

In 2015, LSTC introduced a new structured ALE (S-ALE) solver option dedicated to solve the subset of ALE problems where a structured mesh is appropriate. As expected, recognizing the logical regularity of the mesh brought a reduced simulation time for the case of identical structured and unstructured mesh definitions. It also comes with a cleaner, conceptually simpler way of model setup.

Since the past year, efforts have been made to expand S-ALE into the 2D territory. This would be beneficial to both users and the developer. Using a single set of coding for both 3D and 2D models would greatly simplify the maintenance. In the meantime, it also provides a unified platform for future developments. As S-ALE 3D coding has been tested intensively during the last five years, we believe the S-ALE 2D solver would be stable and efficient from the beginning. For users, using the same set of keywords in both 3D and 2D applications could reduce the learning time and modeling efforts so they could focus more on the physical part of the modeling.

This article gives a brief description of the S-ALE 2D model setup.

Three step setup

We follow a straight-forward three step setup. First, mesh; secondly, material properties of fluids; thirdly, filling the mesh with fluids. In this section, we describe the three keywords doing these three steps.

1. Mesh generation: *ALE_STRUCTURED_MESH; *ALE_STRUCTURED_MESH_CONTROL_POINTS

In S-ALE, mesh is always rectangular. To determine the mesh layout in the space, we need the following information:

    a. Mesh spacing along three axes (LCIDX,LCIDY,LCIDZ) in 3D; two axes (LCIDX,LCIDY) in 2D.
    b. The origin (NID0), and local coordinate system (LCSID).

Other fields are for identification purpose only and are self-explanatory. MSHID stands for mesh ID; DPID Part ID; NBID and EBID are the IDs of first S-ALE mesh node and element, respectively. TDEATH is to the "death time" for S-ALE mesh. It is to turn off the S-ALE calculation once the

most of fluid loading is applied; and keep the Lagrange model running as the structure deformation is not fully developed yet.

*ALE_STRUCTURED_MESH							
MSHID	DPID	NBID	EBID				TDEATH
CPIDX	CPIDY	CPIDZ	NID0	LCSID			

The CPIDX, CPIDY and CPIDZ are IDs of *ALE_STRUCTURED_MESH_CONTROL_POINTS cards. Each card specifies the mesh spacing along one axis.

*ALE_STRUCTURED_MESH_CONTROL_POINTS							
CPID							
N1		X1		RATIO1			
…		…		…			
Nn		Xn		RATIOn			

In a 2D simulation, the mesh is 2 dimensional so CPIDZ is no longer needed. Nonzero value of CPIDZ would be simply ignored and hence harmless. For 2D Axisymmetric case, special caution needs to be taken when using NID0 (origin shift) and LCSID (mesh rotation) feature, to make sure the created mesh is not in conflict with the assumption that the axis of symmetry is at x=0.

2. Material definitions: *ALE_STRUCTURED_MULTI-MATERIAL_GROUP_?

S-ALE mesh is simply a spatial domain in which fluids flow. In order to let the code know what and how many fluids there are, we need to provide material properties of each fluid and list them all under the card *ALE_STRUCTURED_MULTI-MATERIAL_GROUP. Please note, AMMG stands for ALE multi-material group, a rather alternative and maybe confusing name for "ALE fluid". In this paper we are going to use AMMG and fluid interchangeably.

For 2D, *ALE_STRUCTURED_MULTI-MATERIAL_GROUP also carries another role. It is to specify the nature of the 2D simulation (element formulation), i.e. Plane strain or Axisymmetric. An option string at the end, either PLNEPS or AXISYM does the job. These two options directly correspond to setting elform=13 and elfrom=14 in *SECTION_ALE2D card.

*ALE_STRUCTURED_MULTI-MATERIAL_GROUP_PLNEPS/AXISYM							
AMMGNM1	MID1	EOSID1					PREF1
…	…	…					…
AMMGNMn	MIDn	EOSIDn					PREFn

"AMMGNM" is a name one gives to a AMMG (ALE Multi-Material Group), aka ALE fluid. It is used in other cards, for example, *SET_MULTI-MATERIAL_GROUP_LIST to refer to that AMMG. "MID" and "EOSID" are the material ID and EOS ID, respectively.

"PREF" is to describe the reference pressure or "base pressure" of that fluid. This might be somewhat new to our typical users from solids background. Pressure of a solid material, if not preloaded, always starts from zero. In such case, its reference pressure or base pressure, is zero. But most fluids have non-zero reference pressure. For example, air has a base pressure of 101325 Pa (1 bar atmospheric pressure). Traditionally this reference pressure is prescribed using the field "PREF" in *CONTROL_ALE card. The new *ALE_STRUCTURED_MULTI-MATERIAL_GROUP has a design to allow each AMMG to have its own reference pressure. The author believes this added flexibility could be proven very useful in certain applications.

3. Volume Filling: *ALE_STRUCTURED_MESH_VOLUME_FILLING

We created a rectangular 2D S-ALE mesh in step 1 and came up multiple fluids (AMMGs) definitions in step 2. We now fill the mesh with those fluids. We do that by specifying which fluid occupies either inside or outside of certain geometry. These geometries could be simple shapes like sphere, plane, box, cylinder. Or it could be user-defined complex shape like structure surfaces.

This is done by using the keyword "*ALE_STRUCTURED_MESH_VOLUME_FILLING". The "volume filling" process typically is done through multiple "tasks", each task by a separate keyword.

*ALE_STRUCTURED_ MESH_VOLUME_ FILLING							
MSHID		AMMGTO		NSAMPLE			VID
GEOM	IN/OUT	E1	E2	E3	E4		

MSHID is the S-ALE mesh ID, defined in *ALE_STRUCTURED_MESH card. And AMMGTO is the name of ALE fluid to be filled, defined in *ALE_STRCUTURED_MULTI-MATERIAL_GROUP. VID is to prescribe the initial velocity of that fluid, if any. And NSAMPLE is default to 3 which means that one ALE cell is divided into 7x7x7 (7=2*3+1) sub-cells in 3D and 7x7 in 2D and each sub-cell is checked to see if it is inside/outside.

It supports 5 basic geometries: Plane, Cylinder, Ellipsoid, Box with indices and Box with coordinates. And E1-E4 are used to provide information of these geometries. For complicated geometries, we need users to provide us with a segment set (or something we could internally convert to a segment set). The assumption is that all segment normals are consistent; and those normals point to the "inside". For convenience, we provide a "IN/OUT" flag for an easier flip.

Boundary/Initial Conditions and *CONTROL_ALE

Boundary Conditions. The most used type of boundary condition is single point constraint (SPC). It is to constrain motion at a point (node) along certain direction. Traditionally, we use *BOUNDARY_SPC to apply SPC constraints in S-ALE. In R12.1 (Ansys 2021R2), we added a new "macro-like" keyword "*BOUNDARY_SALE_MESH_FACE" to make this process more user-friendly.

*BOUNDARY_SALE_MESH_FACE							
OPTION	MSHID	-X	+X	-Y	+Y	-Z	+Z
SYM	1			1			
NOFLOW	1					1	1
NONREFL	1	1	1		1		

Three options, SYM, NOFLOW and FIXED, are for SPC constraint. SYM and NOFLOW, are doing the same thing. That is to constrain the flow perpendicular to the plane. We have both options available, simply to provide a one-to-one match between the options and real physical scenarios. FIXED is to fix all nodal motions at that face.

The other one, NONREFL, is to apply non-reflecting boundary at S-ALE mesh face(s). Internally this option is translated to *BOUNDARY_NON_REFLECTING + *SET_SEGMENT.

Initial conditions. IC setup is relatively simple. Typically it is only to assign some initial velocity to some nodes. Most commonly, initial velocities are applied simultaneously with volume filling process, through the *ALE_STRUCTURED_MESH_VOLUME_FILLING card. Like the following:

```
*ALE_STRUCTURED_MESH_VOLUME_FILLING
$#   mshid         -     ammgto        -     nsample        -         -         vid
         1                    plate                                              1
$#    geom    in/out      boxid
BOXCOR              0         1
```

And the initial velocity (vx=-61.631,vy=208.06) is prescribed by using the *DEFINE_VECTOR card.

```
*DEFINE_VECTOR
$#     vid        xt        yt        zt        xh        yh        zh       cid
         1   -61.631    208.06       0.0       0.0       0.0       0.0         0
```

Control Card. *CONTROL_ALE is used to set up certain global control parameters in a S-ALE simulation. The only relevant parameters are: NADV and METH. METH=1/2 controls the order of advection scheme, 1st or 2nd order. NADV could be adjusted to increase the interval (in number of cycles) between two advections. It is to provide users with a way to shorten the running time for problems in which deformation is small compared to element size in one cycle. Then we could wait until the element is severely deformed and then perform an advection (remapping).   All other parameters most cases the user could safely ignore.

```
*CONTROL_ALE
$#     dct      nadv      meth      afac      bfac      cfac      dfac      efac
                   1         1 -1.000000
$#   start       end     aafac     vfact      prit       ebc      pref     idebc
```

The parameter PREF was to set the reference pressure. However, S-ALE came up with a better design to allow users to prescribe reference pressure for each ALE fluid (AMMG) through "PREFn"

in *ALE_STRUCTURED_MULTI-MATERIAL_GROUP.   PREF will still be kept there for backward compatibility but we strongly recommend setting reference pressure through the *ASMMG card.

Example

Now let us use a simple example to illustrate the 2D S-ALE setup. We have a rigid body wedge composed of beam elements slamming into water as the figure shown below. It is a 2D plane strain model. The unit used is mm-ton-s. Pressure unit is Mpa.
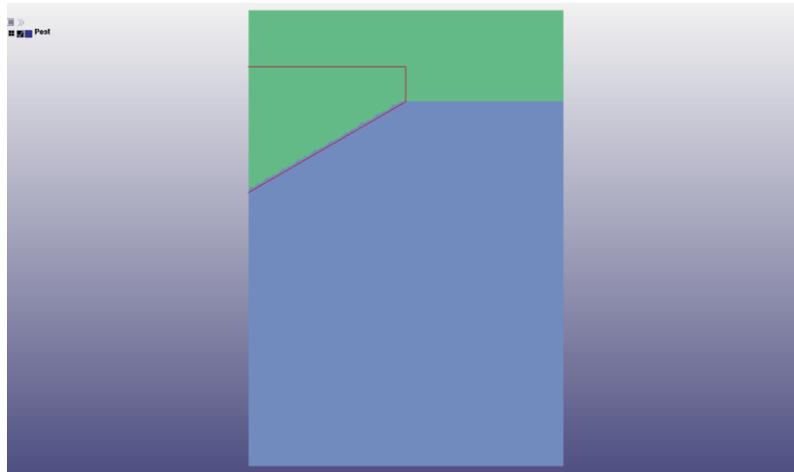
Step 1: To construct a mesh spans (0,-433.013) to (500,288.675) with 100x150 elements.

```
*ALE_STRUCTURED_MESH
$    mshid       pid      nbid      ebid
         1        11    100001    100001
$    cpidx     cpidy     cpidz      nid0      lcsid
      1001      1002      1003
*ALE_STRUCTURED_MESH_CONTROL_POINTS
      1001
$               x1                x2                x3                x4
               1               0.0
             101             500.0
*ALE_STRUCTURED_MESH_CONTROL_POINTS
      1002
$               x1                x2                x3                x4
               1          -433.013
             151           288.675
*ALE_STRUCTURED_MESH_CONTROL_POINTS
      1003
$               x1                x2                x3                x4
               1              -0.5
               2               0.5
```

Here CPIDZ is harmless. All nodal coordinates will be initialized with z=0. We will see its purpose later.

Step 2: Set up ALE multi-material Groups (AMMGs or ALE fluids). There are totally 2 AMMGs defined. First is "water" by *MAT_NULL (MID=4) and *EOS_LINEAR_POLYNOMIAL (EOSID=4).

The second is "air" by * MAT_NULL (MID=5) and *EOS_LINEAR_POLYNOMIAL (EOSID=5). Their materials properties are given as follows.

```
*MAT_NULL
$#      mid          ro          pc          mu       terod       cerod          ym          pr
          4  1.026e-9         0.0         0.0         0.0         0.0         0.0         0.0
*EOS_LINEAR_POLYNOMIAL
$#    eosid          c0          c1          c2          c3          c4          c5          c6
          4         0.0         2.3         0.0         0.0         0.0         0.0         0.0
$#       e0          v0
         0.0         0.0
*MAT_NULL
$#      mid          ro          pc          mu       terod       cerod          ym          pr
          5 1.293e-12         0.0         0.0         0.0         0.0         0.0         0.0
*EOS_LINEAR_POLYNOMIAL
$#    eosid          c0          c1          c2          c3          c4          c5          c6
          5         0.0         0.0         0.0         0.0         0.4         0.4         0.0
$#       e0          v0
         0.0         0.0
```

Here for simplicity, we assigned initial pressure to be zero for both air and water. In problems where the air compressibility becomes important, we could prescribe the initial pressure of 1 bar by assigning e0=0.101325/0.4=0.258125. If we do that, please do not forget to set the reference pressure of water to 1 bar by making PREF=0.101325 in *ALE_STRUCTURED_MULTI-MATERIAL_GROUP (*ASMMG) otherwise water pressure is not well balanced.

Now construct the ALE fluids by using *ALE_STRUCTURED_MULTI-MATERIAL_GROUP. And specify the nature of the simulation to 2D plane strain by adding option _PLNEPS at the end.

```
*ALE_STRUCTURED_MULTI-MATERIAL_GROUP_PLNEPS
$     name      mid    eosid                                                        pref
    water         4        4
    air           5        5
```

Step 3: Volume Filling the initial S-ALE mesh. By *ALE_STRUCTRUED_MESH_VOLUME_FILLING.

First, fill all with "water".

```
*ALE_STRUCTURED_MESH_VOLUME_FILLING
$#   mshid          -      ammgto          -    nsample          -          -         vid
         1                  water
$#    geom     in/out
ALL
```

Next, switch above the plane to "air".

```
*ALE_STRUCTURED_MESH_VOLUME_FILLING
$#   mshid          -      ammgto          -    nsample          -          -         vid
         1                  air
$#    geom     in/out      nid1       nid2
 PLANE                    10001      10002
*NODE
$#   nid               x               y               z        tc        rc
   10001             0.0         144.338
```

```
      10002                 0.0             145.338
```

Finally, we switch the fluid inside the wedge to "air".

```
*ALE_STRUCTURED_MESH_VOLUME_FILLING
$#   mshid          -     ammgto        -    nsample        -          -        vid
         1                ammgto
                          air
$#    geom     in/out    ssetid
 SEGSET             1         1
```

Boundary Conditions and *CONTROL_ALE card

```
*BOUNDARY_SALE_MESH_FACE
$   option     mshid         -x        +x        -y        +y        -z        +z
    NOFLOW         1          1         1         1         1         1         1
```

Here we also set the SPC constraints at -z and +z mesh faces. It is not necessary but generally harmless. We will see why we did that in a minute.

```
*CONTROL_ALE
$#      dct       nadv      meth      afac      bfac      cfac      dfac      efac
                             1
$#    start        end     aafac     vfact      prit       ebc      pref   nsidebc
```

Remember how we want our users to be minimalist on *CONTROL_ALE?

FSI Setup

In the next R12.1 (Ansys 2021R2) release, *ALE_STRUCTURED_FSI card becomes fully functional with added 2D support. And here we are using this much simpler, stabler card embedded with a better leakage control algorithm.

```
*ALE_STRUCTURED_FSI
$#   slave     master      sstyp     mstyp                                    mcoup
         1         11          2         1                     4             -123
$#   start        end       pfac                           flip
       0.0        0.0         -1
*SET_MULTI_MATERIAL_GROUP_LIST
$#  ammsid
       123
$# ammgid1   ammgid2   ammgid3   ammgid4   ammgid5   ammgid6   ammgid7   ammgid8
    water
*DEFINE_CURVE
$#    lcid
         1
$#               a1                o1
                0.0               0.0
                1.0               1.0
```

Simply put, what we did in the above cards is as follows:

1. Couple a segment set (sstyp=2) with id 1 (slave=1) to "water" (mcoup=-123) in the S-ALE mesh part (mstyp=1) with id 11 (master=11).

2. Specify the couple stiffness to use the load curve id 1 (PFAC=-1). This load curve establishes a linear relationship between penetration and penalty spring pressure. It comes with two points. The first is always (0,0) which means no penalty pressure is given when the penetration is zero. The second point we recommend (0.1xdl, p_max). This is to tell the FSI algorithm to apply the p_max on coupling surface when fluids penetration is around 1/10 of the S-ALE element size. P_max is the maximum impact pressure that users need to come up with their own guess/estimate. The load curve used here instructs FSI to apply a penalty pressure of 10 bars at a penetration of 1 mm. We did not follow the rule here only because the toy nature of this simple model.

Below is the water and wedge at t=0., t=7.5ms and t=15ms. The input deck is available at https://ftp.lstc.com/anonymous/outgoing/hao/sale/models_R121/slam_wedge/. It took only 2 seconds to run this model with my 16 core machine.
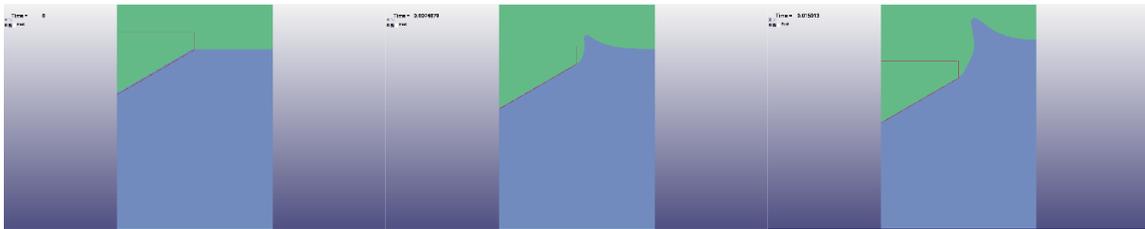
Axisymmetric and 3D models

Here we show how easy we could switch between Plane Strain, Axisymmetric and 3D simulations. First let us change this plane strain model to an axisymmetric one. All we need to do is to replace PLNEPS by AXISYM in the *ASMMG card.

```
*ALE_STRUCTURED_MULTI-MATERIAL_GROUP_AXISYM
$     name       mid     eosid                                        pref
    water           4         4
   air              5         5
```

For Lagrange part of the model, we need to change the beam element formulation from 7 (plane strain) to 8 (axisymmetric). And that is it!

```
*SECTION_BEAM
$#    secid      elform
         1           8
$#       t1         t2         t3         t4
       1.0        1.0        1.0        1.0
```

Now we change it to a 3D model. All we need to do is to strike out _PLNEPS from the *ASMMG card. Remember we had a CPIDZ definition in step 1 and also set the SPC constraints at -z and z faces? Those two are harmless in a 2D simulation but in case you want to do a 3D comparison then it would be really handy.

```
*ALE_STRUCTURED_MULTI-MATERIAL_GROUP
$     name       mid     eosid                                        pref
```

```
     water               4          4
     air                 5          5
```
Of course, for LAG part, we need to prepare a different set of mesh with different nodes and elements defined. The model input deck contains both 2D and 3D Lagrange mesh and one could include the corresponding mesh files.

```
*INCLUDE
$mesh_wedge_2d.k
mesh_wedge_3d.k
```

3D simulation is a little bit more costly – 5 seconds on my 16 core machine. So we urge the reader to download the test case input deck and play with the switching between plane strain, axisymmetric and 3D simulations. We promise it is going to be fast and easy!

Below figure compares FSI pressure between plane strain and 3D simulations. We could see these two curves almost on top of each other. This validates the newly developed 2D S-ALE works as expected.
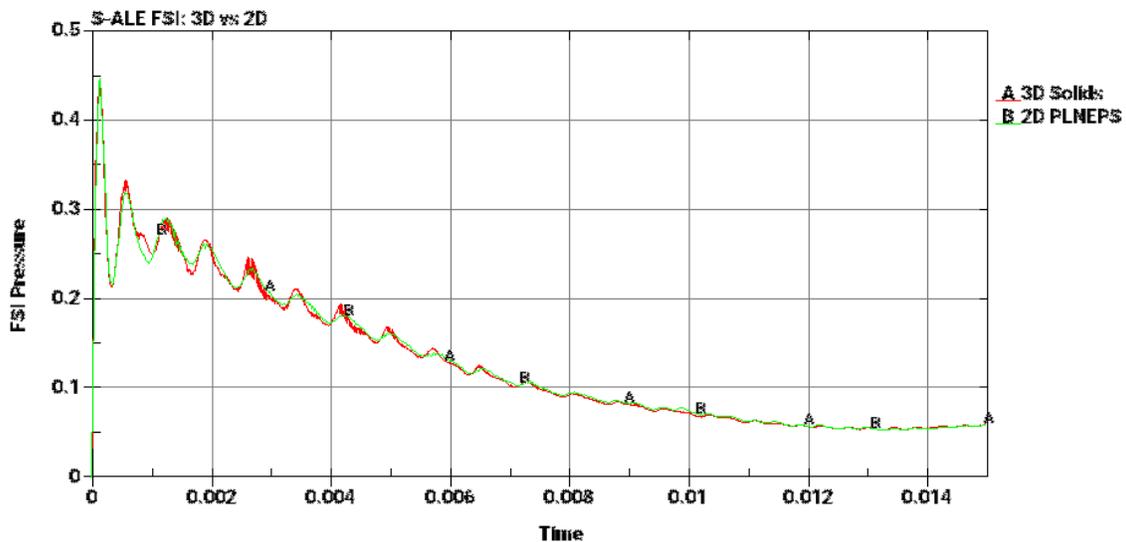
Ending Remarks

LS-DYNA ALE module has been known for its steep learning curve. Partially it was because setting up Eulerian models are intrinsically different from Lagrange models. But the design of ALE keyword cards, for sure, has caused quite a lot of confusions among our users, new and experienced.

To prompt LS-DYNA ALE usages, Structured ALE solver introduced a new, user-friendly, streamlined three-step setup. We hope this effort could help users, new or old, to perform their work more efficiently and smoothly.