*SET_NODE_{OPTION1}_{OPTION2}

For *OPTION1* the available options are:

<BLANK>

LIST

COLUMN

LIST_GENERATE

LIST_GENERATE_INCREMENT

GENERAL

LIST_SMOOTH

For *OPTION2* the available option is:

COLLECT

The LIST option generates a set for a list of node IDs. The LIST_GENERATE and LIST_GENERATE_INCREMENT options will generate block(s) of node IDs between a starting ID and an ending ID. An arbitrary number of blocks can be specified to define the node set. The option LIST_SMOOTH is used to define a local region on a distorted tooling mesh to be smoothed, see **Local smoothing of tooling mesh** in *INTERFACE_COMPENSATION_NEW. The COLUMN option is for setting nodal attributes, which pass data to other keyword cards, on a node-by-node basis.

Purpose: Define a nodal set with some identical or unique attributes.

Card 1	1	2	3	4	5	6	7	8
Variable	SID	DA1	DA2	DA3	DA4	SOLVER		
Type	I	F	F	F	F	A		
Default	none	0.	0.	0.	0.	MECH		
Remark		1	1	1	1	3		

Node ID Cards. This Card 2 format applies to LIST and LIST_SMOOTH keyword options. Additionally, it applies to the case of an unset (<BLANK>) keyword option. Set one value per node in the set. Include as many cards as needed. This input ends at the next keyword ("*") card.

Card 2	1	2	3	4	5	6	7	8
Variable	NID1	NID2	NID3	NID4	NID5	NID6	NID7	NID8
Type	I	I	I	I	I	I	I	I

Node ID with Column Cards. This Card 2 format applies to the COLUMN keyword option. Include one card per node in the set. Include as many cards as needed. This input ends at the next keyword ("*") card.

Card 2	1	2	3	4	5	6	7	8
Variable	NID	A1	A2	A3	A4			
Type	I	F	F	F	F			
Remark		2	2	2	2			

Node ID Range Cards. This Card 2 format applies to the GENERATE keyword option. Set one pair of BNBEG and BNEND values per block of nodes. Include as many cards as needed. This input ends at the next keyword ("*") card.

Card 2	1	2	3	4	5	6	7	8
Variable	B1BEG	B1END	B2BEG	B2END	B3BEG	B3END	B4BEG	B4END
Type	I	I	I	I	I	I	I	I

Node ID Range with Increment Cards. This Card 2 format applies to the LIST_GENERATE_INCREMENT keyword option. For each block of nodes add one card to the deck. This input ends at the next keyword ("*") card.

Card 2	1	2	3	4	5	6	7	8
Variable	BBEG	BEND	INCR					
Type	I	I	I					

Generalized Node ID Range Cards. This Card 2 format applies to the GENERAL keyword option. Include as many cards as needed. This input ends at the next keyword ("*") card.

Card 2	1	2	3	4	5	6	7	8
Variable	OPTION	E1	E2	E3	E4	E5	E6	E7
Type	A	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
SID	Set identification. All node sets should have a unique set ID.
DA1	First nodal attribute default value, see remark 1 below.
DA2	Second nodal attribute default value
DA3	Third nodal attribute default value
DA4	Fourth nodal attribute default value
SOLVER	Name of solver using this set (MECH, CESE, etc.)
NID*i*	Node ID *i*
NID	Nodal ID
A1	First nodal attribute, see remark 2 below.
A2	Second nodal attribute
A3	Third nodal attribute
A4	Fourth nodal attribute

VARIABLE	DESCRIPTION
B*n*BEG	First node ID in block *n*.
B*n*END	Last node ID in block *n*. All defined ID's between and including B*n*BEG to B*n*END are added to the set. These sets are generated after all input is read so that gaps in the node numbering are not a problem. B*n*BEG and B*n*END may simply be limits on the ID's and not nodal ID's.
BBEG	First node ID in block.
BEND	Last node ID in block.
INCR	Node ID increment. Node IDs BBEG, BBEG + INCR, BBEG + 2 × INCR, and so on through BEND are added to the set.
OPTION	Option for GENERAL. See table below.
E1, …, E7	Specified entity. Each card must have the option specified. See table below.

The General Option:

The "OPTION" column in the table below enumerates the allowed values for the "OPTION" variable in Card 2 for the GENERAL option. Likewise, the variables E1, …, E7 refer to the GENERAL option Card 2.

Each of the following operations accept up to 7 arguments, but they may take fewer. Values of "En" left unspecified are ignored.

OPTION	DESCRIPTION
ALL	All nodes will be included in the set.
NODE	Nodes E1, E2, E3, … will be included.
DNODE	Nodes E1, E2, E3, … previously added will be excluded.
PART	Nodes of parts E1, E2, E3, … will be included.
DPART	Nodes of parts E1, E2, E3, … previously added will be excluded.
BOX	Nodes inside boxes E1, E2, E3, … will be included. (see *DEFINE_ BOX)
DBOX	Nodes inside boxes E1, E2, E3, … previously added will be excluded.

OPTION	DESCRIPTION
VOL	Nodes inside contact volumes E1, E2, E3, … will be included. (see *DEFINE_CONTACT_VOLUME)
DVOL	Nodes inside contact volumes E1, E2, E3, … previously added will be excluded.
SET_*XXXX*	Include nodal points of element sets defined by SET_*XXXX*_LIST, where *XXXX* could be SHELL, SOLID, BEAM, TSHELL and SPRING
SALECPT	Nodes inside a box in Structured ALE mesh. E1 here is the S-ALE mesh ID (MSHID). E2, E3, E4, E5, E6, E7 correspond to XMIN, XMAX, YMIN, YMAX, ZMIN, ZMAX. They are the minimum and the maximum nodal indices along each direction in S-ALE mesh. This option is only to be used for Structured ALE mesh and should not be used in a mixed manner with other "_GENERAL" options. Please refer to *ALE_STRUCTURED_MESH_CONTROL_POINTS and *ALE_STRUCTURED_MESH_CONTROL for more details.
SALEFAC	Nodes on the face of Structured ALE mesh. E1 here is the S-ALE mesh ID (MSHID). E2, E3, E4, E5, E6, E7 correspond to -X, +X, -Y, +Y, -Z, +Z faces. Assigning 1 to these 6 values would include all the surface segments at these faces in the segment set. This option is only to be used for Structured ALE mesh and should not be used in a mixed manner with other "_GENERAL" options. Please refer to *ALE_STRUCTURED_MESH_CONTROL_POINTS and *ALE_STRUCTURED_MESH_CONTROL for more details.

Remarks:

1. Nodal attributes can be assigned to pass data to other keywords. For example, for contact option, *CONTACT_TIEBREAK_NODES_TO_SURFACE the attributes are:

   DA1 = NFLF ⇒ Normal failure force,

   DA2 = NSFL ⇒ Shear failure force,

   DA3 = NNEN ⇒ Exponent for normal force,

   DA4 = NMES ⇒ Exponent for shear force.

2. The default nodal attributes can be overridden on these cards; otherwise, A1=DA1, etc.

3. This field is used by a non-mechanics solver to create a set defined on that solver's mesh. By default, the set refers to the mechanics mesh.

4. The option *SET_NODE_LIST_SMOOTH is used for localized tooling surface smoothing, and is used in conjunction with keywords *INTERFACE_ COMPENSATION_NEW_LOCAL_SMOOTH, *INCLUDE_COMPENSATION_ ORIGINAL_RIGID_TOOL, and *INCLUDE_COMPENSATION_NEW_RIGID_ TOOL. This option is available in R6 Revision 73850 and later releases