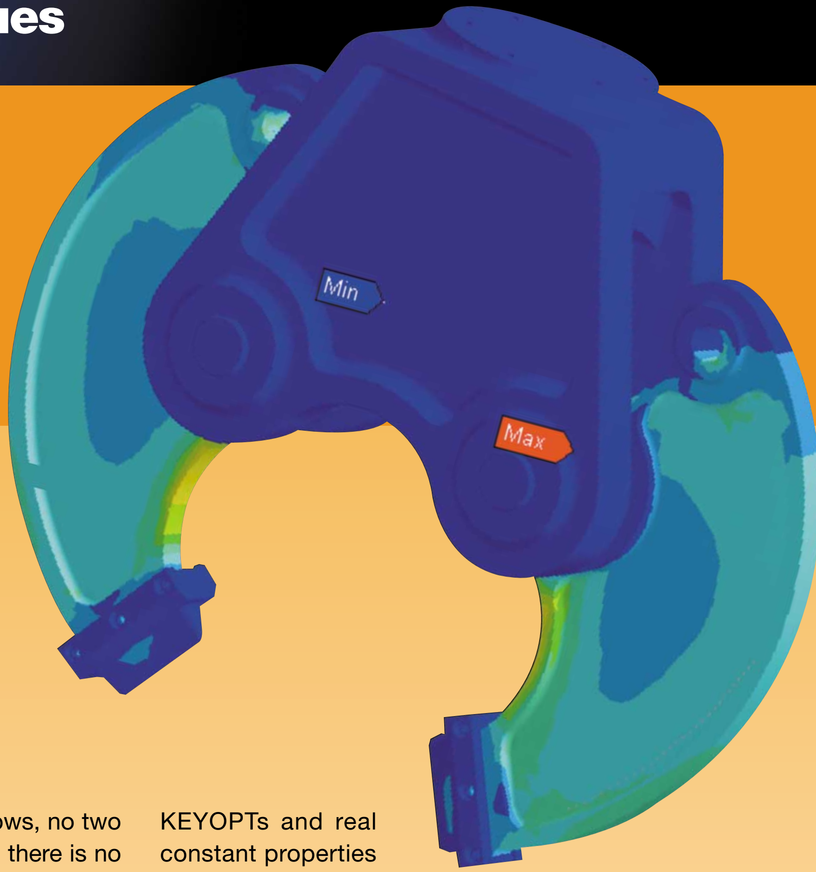
Contact Defaults in Workbench and ANSYS



Intelligent default settings solve common problems fast with minimal user intervention.

By ANSYS, Inc. Technical Support

As every experienced FEA analyst knows, no two contact problems are exactly alike, so there is no silver bullet combination of KEYOPT and real constant settings that will successfully work for all problems. That explains the many features available today within the contact elements. It also explains, in part, the rationale behind the different default settings sometimes found in the different environments. As migration between Workbench and ANSYS environments progresses, it is important for analysts to recognize that, although the contact technology used in both of these environments is exactly the same, some of the default KEYOPT and real constant settings are not.

Tables 1 and 2 summarize all surface-to-surface contact element (CONTA171–174) KEYOPTs and real constant properties with their respective default settings in each environment. Those that have different defaults in the different environments are highlighted in ***bold italic***.

KEYOPT(1): Select Degrees of Freedom (DOF)

This option gives you the freedom to assign the contact DOF set consistent with the physics of the underlying elements. ANSYS surface-to-surface contact technology offers an impressive combination of structural, thermal, electric and magnetic capabilities. When building pairs through the ANSYS environment with traditional ANSYS Parametric Design Language (APDL), users must

Table 1: 8.0 Default Contact KEYOPTs

KEYOPTs	Description	ANSYS APDL	Contact Wizard	Workbench Default Linear (bonded, no sep)	Workbench Default Nonlinear (standard, rough)
1	***Selects DOF***	***manual***	***automatic***	***automatic***	***automatic***
2	***Contact algorithm***	***Aug Lagrange***	***Aug Lagrange***	***pure penalty***	***pure penalty***
3	Stress state when super element is present	no super elem	no super elem	n/a	n/a
4	Location of contact detection point	gauss	gauss	gauss	gauss
5	CNOF/ICONT adjustment	no adjust	no adjust	no adjust	no adjust
6	(blank)				
7	Element level time increment control	no control	no control	no control	no control
8	Asymmetric contact selection	no action	no action	no action	no action
9	***Effect of initial penetration or gap***	***include all***	***include all***	***exclude all***	***include all/ ramped***
10	***Contact stiffness update***	***btwn loadsteps***	***btwn substps***	***btwn loadsteps***	***btwn loadsteps***
11	Beam/shell thickness effect	exclude	exclude	exclude	exclude
12	***Behavior of contact surface***	***standard***	***standard***	***bonded***	***n/a***

Table 2: 8.0 Default Contact Real Constants

Real Constants		Description	ANSYS APDLContact	Wizard	Workbench
No.	Name				
1	R1	Target circle radius	0	n/a	n/a
2	R2	Superelement thickness	1	1	n/a
3	***FKN***	***Normal penalty stiffness factor***	***1***	***1***	***Note 1***
4	FTOLN	Penetration tolerance factor	0.1	0.1	0.1
5	ICONT	Initial contact closure	0	0	0
6	PINB	Pinball region	note 2	note 2 n	ote 2
7	PMAX	Upper limit of initial penetration	0	0	0
8	PMIN	Lower limit of initial penetration	0	0	0
9	TAUMAX	Maximum friction stress	1.00E+20	1.00E+20	1.00E+20
10	CNOF	Contact surface offset	0	0	0
11	FKOP	Contact opening stiffness	1	1	1
12	FKT	Tangent penalty stiffness	1	1	1
13	COHE	Contact cohesion	0	0	0
14	***TCC***	***Thermal contact conductance***	***0***	***0***	***Note 3***
15	FHTG	Frictional heating factor	1	1	1
16	SBCT	Stefan-Boltzmann constant	0	0	n/a
17	RDVF	Radiation view factor	1	1	n/a
18	FWGT	Heat distribution weighing factor	0.5	0.5	0.5
19	ECC	Electric contact conductance	0	0	n/a
20	FHEG	Joule dissipation weighting factor	1	1	n/a
21	FACT	Static/dynamic ratio	1	1	1
22	DC	Exponential decay coefficient	0	0	0
23	SLTO	Allowable elastic slip	1%	1%	1%
25	TOLS	Target edge extension factor	note 4	note 4	note 4
26	MCC	Magnetic contact permeance	0	0	n/a

Notes:
1. *FKN = 10 if only linear contact is active (bonded, no sep). If any nonlinear contact is active, all regions will have FKN = 1 (including bonded, no sep).*
2. *Depends on contact behavior, rigid vs. flex target, KEYOPT (9) and NLGEOM ON/OFF.*
3. *Calculated as a function of highest conductivity and overall model size.*
4. *10% of target length for NLGEOM,OFF. 2% of target length for NLGEOM,ON.*

set this option manually. The default will always be KEYOPT(1) =0 (for UX,UY). When building contact pairs in the ANSYS environment using the contact wizard, KEYOPT(1) is set automatically according to the DOF set of the underlying element. In Workbench, this option also is set automatically, depending on the underlying element DOF set.

KEYOPT(2): Contact Algorithm

ANSYS contact technology offers many algorithms to control how the code enforces compatibility at a contacting interface.

The penalty method (KEYOPT(2) =1) is a traditional algorithm that enforces contact compatibility by using a contact “spring” to establish a relationship between the two surfaces. The spring stiffness is called the penalty parameter or, more commonly, the contact stiffness. The spring is inactive when the surfaces are apart (open status), and becomes active when the surfaces begin to interpenetrate.

The augmented Lagrange method (KEYOPT(2) = 0) uses an iterative series of penalty methods to enforce contact compatibility. Contact tractions (pressure and friction stresses) are augmented during equilibrium iterations so that final penetration is smaller then the allowable tolerance. This offers better conditioning than the pure penalty method and is less sensitive to magnitude of contact stiffness used, but may require more iterations than the penalty method.

The Multi-Point Constraint (MPC) Method (KEYOPT(2) = 2) enforces contact compatibility by using internally generated constraint equations to establish a relationship between the two surfaces. The DOFs of the contact surface nodes are eliminated. No normal or tangential stiffness is required. For small deformation problems, no iterations are needed in solving system equations. Since there is no penetration or contact sliding within a tolerance, MPC represents "true linear contact" behavior. For large deformation problems, the MPC equations are updated during each iteration. This method applies to bonded surface behavior only. It is also useful for building surface constraint relationships similar to CERIG and RBE3. MPC is available as a standard option when modeling bonded contact in both ANSYS and Workbench environments.

The Pure Lagrange multiplier method (KEYOPT(2) = 3) adds an extra degree of freedom (contact pressure) to satisfy contact compatibility. Pure Lagrange enforces near-zero penetration with pressure DOF. Unlike the penalty and augmented Lagrange algorithms, it does not require a normal contact stiffness. Pure Lagrange does require a direct solver, can be more computationally expensive and can have convergence difficulties related to overconstraining, but it is a very useful algorithm when zero penetration is critical. It also can be combined with the penalty algorithm in the tangential direction (KEYOPT(2) = 4), when zero penetration is critical, and friction is also present.

The ANSYS environment uses the augmented Lagrange by default. The Workbench environment currently uses the penalty method, but the default can be changed via the Options Menu at 8.1. MPC is available as a standard alternative in both environments. The Pure Lagrange options are available in ANSYS, but can be accessed in Workbench via the pre-processor command builder. At version 8.1, Pure Lagrange is available in the Workbench environment. Table 3 summarizes all the algorithms with pros and cons of each.

KEYOPT(9): Effect of Initial Penetration or Gap

Properly accounting for or controlling interferences and gaps can sometimes be the difference between success and failure in simulating a complicated contact relationship. There are several contact options available to control how the code accounts for initial interference or gap effects:

(0) ***Include everything:*** Include an initial interference from the geometry and the specified offset (if any).

(1) ***Exclude everything:*** Ignore all initial-interference effects.

(2) **Include with ramped effects:** Ramp the interference to enhance convergence.

(3) ***Include offset only:*** Base initial interference on specified offset only.

(4) ***Include offset only w/ ramp:*** Base initial interference on specified offset only, and ramp the interference effect to enhance convergence.

Table 3: Contact Algorithms

Algorithm	Pros	Cons	When to Use
Pure Penalty	Offers easiest convergence in least number of iterations	Requires contact stiffness and allowance for some finite penetration	Helpful when contact convergence is a challenge and minimal penetration is acceptable (Default in Workbench)
Augmented	Minimizes penetration; better conditioning than penalty; less sensitive to contact stiffness	Might require more iterations	The default for surf-to-surf and node-to-surf in ANSYS, as it has proven to produce the best quality results in the most common applications (Default in ANSYS)
Pure Lagrange	Offers near-zero penetration; zero elastic slip (no contact stiffness required)	Might require more iterations; might also require adjustment to chatter control parameters unique to this algorithm; can produce overconstraints in model	When zero penetration is critical
Pure Lagrange on Normal; Penalty on Tangent	Same as Pure Lagrange, plus simulation of friction is handled most efficiently	Same as Pure Lagrange	When zero penetration is critical and friction is present
Multipoint Constraint (MPC)	More efficient than traditional bonded contact; offers contact betweenmixed element types; offers CERIG RBE3 type constraints	Can produce overconstraints in model	Recommended for large bonded contact models to enhance run time and for contact between mixed element types and surface constraint applications

In ANSYS, the default KEYOPT(9) = 0 is to include everything. In Workbench, the default is to exclude everything (1) when linear contact (bonded, no separation) is defined and include with ramped effects (2) when nonlinear contact (frictional, frictionless, rough) is defined.

KEYOPT(10): Contact Stiffness Update

When using the penalty and/or augmented Lagrange method, contact stiffness has long been recognized as a critical property that influences both accuracy and convergence. Too high a stiffness will ultimately lead to convergence difficulty; too low a stiffness will result in over-penetration and an inaccurate assessment of surface pressures and stresses at the interface. In an effort to arrive at a good balance between these extremes, automatic stiffness updating between loadsteps (KEYOPT(10) = 0) and substeps (KEYOPT(10) = 1), or between iterations (KEYOPT(10) = 2) was introduced as an enhancement to traditional trial-and-error methods.

In ANSYS, when contact is built via APDL, the default is to update stiffness between loadsteps. In ANSYS, when contact is built via the Wizard, the default has been changed to update between substeps. This is considered to produce the most robust contact simulation in most cases. In Workbench, the default behavior is still between loadsteps, but the default can be changed via the Option Menu at Version 8.1. These defaults may change in future releases as further enhancements are made.

KEYOPT(12): Behavior of Contact Surface

ANSYS contact technology offers a rich library of surface behavior options to simulate every possible situation. These options are as follows:

(0) ***Standard:*** (Referred to as "Frictionless" or "Frictional" in Workbench) normal contact closing and opening behavior, with normal sticking/sliding friction behavior when nonzero friction coefficient is defined.

(1) ***Rough:*** Normal contact closing and opening behavior, but no sliding can occur (similar to having an infinite coefficient of friction).

(2) ***No Separation:*** Target and contact surfaces are tied once contact is established (sliding is permitted). This is not available as a standard option in Work-bench, but can be accessed via the pre-processor command builder.

(3) ***Bonded:*** Target and contact surfaces are "glued" once contact is established.

(4) ***No Separation (always):*** (Referred to simply as "No Separation" in Workbench) Any contact detection points initially inside the pinball region or that come into contact are tied in the normal direction (sliding is permitted).

(5) ***Bonded Contact (always):*** (Referred to simply as "Bonded" in Workbench) Any contact detection points initially inside the pinball region or that come into contact are bonded. (Design-space Default)

(6) ***Bonded Contact (initial contact):*** Bonds surfaces ONLY in initial contact, initially open surfaces will remain open. This is not available as a standard option in Workbench, but can be accessed via the pre-processor command builder.

The default surface behavior in ANSYS is nonlinear "standard" for simulating the most general normal contact closing and opening behavior, with normal sticking/sliding friction. In Workbench, the default behavior (which can be changed via the Options Menu at Version 8.1), set up with automatic contact detection to simulate an assembly, is linear Bonded Contact (Always).

Real Constant(3): Normal Penalty Stiffness Factor (FKN)

Users control the initial contact stiffness used by multiplying the calculated value by a factor, FKN. The default value for FKN used in ANSYS (APDL or Wizard) is 1.0. In Workbench, FKN = 10 if only linear contact is active (bonded or no separation). If any nonlinear contact is active, all regions will have FKN = 1 (including bonded and no separation).

Real Constant(14): Thermal Contact Conductance (TCC)

This constant dictates the thermal resistance across the interface of contacting bodies in applications involving thermal analysis. The default value in ANSYS for TCC is zero (perfect insulator). In Workbench, the default is automatically calculated as a function of the highest thermal conductivity of the contacting parts and the overall model sizethus essentially modeling perfect thermal contact. ■