AEROSPACE,
DEFENCE & RAILWAYS

The New Contact with Finite Friction Feature in Creo Simulate 3.0

Theory and Application
Comparison with the Friction-Free and Infinite Friction Contact Models

8th SAXSIM, TU Chemnitz, 22.03.2016

Rev. 1.1 | 24.03.2016

Dr.-Ing. Roland Jakel

altran

At a glance: Altran, a global leader

20+

Countries

23,000+

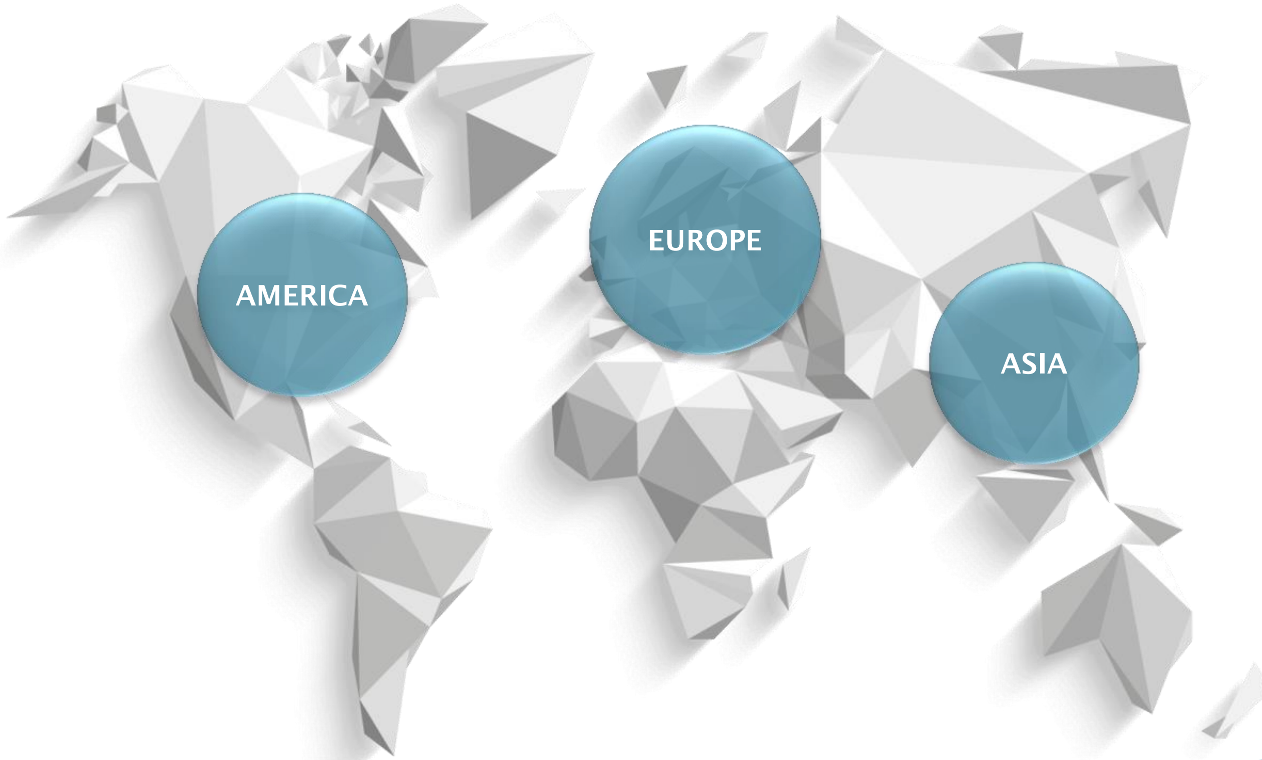
Innovation Makers

5

Industries

€ 1,756m

2014 Revenues

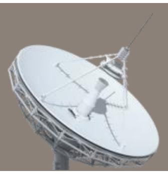


Our Customers

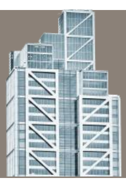
AUTOMOTIVE, INFRASTRUCTURE & TRANSPORTATION

AEROSPACE, DEFENCE & RAILWAYS

ENERGY, INDUSTRY & LIFE SCIENCES



TELECOMS & MEDIA



FINANCIAL SERVICES & PUBLIC SECTOR

Table of Contents (1)

Table of Contents (2)

Part A: Theory & Software Functionality

1. Introduction

1.1 A short overview about contact functionalities in the code

History of contact functionality in Pro/MECHANICA & Creo Simulate

Definition of analysis types often used in this presentation:

- SDA – Small Displacement Analysis, means
  - equilibrium of forces is always done at the non-deformed structure
  - in displacement/rotation analysis, angular functions are linearized, so replaced by the angle itself ($\sin \alpha \approx \tan \alpha \approx \alpha$, valid for small α only)
- LDA – Large Displacement Analysis, means
  - equilibrium of forces is applied iteratively at the deformed structure, until the balanced state is obtained
  - in displacement/rotation analysis, the accurate angular functions are used

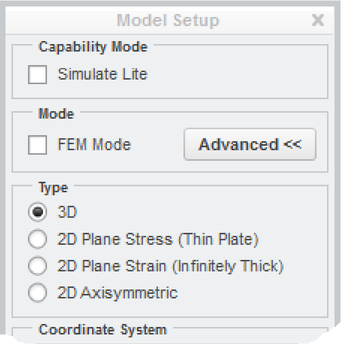
Contact model:	Friction-free	Infinite friction	Finite friction
Introduction with:	Since the nineties	Wildfire 4.0 (2008)	Creo 3.0 (07/2014)
SDA support:	Since introduction of the functionality	Since introduction of the functionality	No, only available in LDA
LDA support:	Since Creo 1.0	Since Creo 1.0	Yes, since intro-duction (Creo 3.0)
Combination with nonlinear material (hyperelastic, plastic):	Since Creo 1.0 (before only linear material)	Since Creo 1.0 (before only linear material)	Yes, since introduction
Combination with snap through (nonlinear stability, requires LDA):	Not supported	Not supported	Not supported

1. Introduction

1.2 Repetition of the contact functionalities implemented until Creo Simulate 2.0

When performing a contact analysis in Creo Simulate, some – but not all – limitations existing in pre-Creo releases have been removed:

- Contact is not limited to SDA any longer as described in [1], also LDA is supported!
- Used material may not only be linear elastic, but also elasto-plastic or hyperelastic
  - Elastoplastic material in Simulate requires use of SDA for small strain and LDA for finite strain plasticity theory, see [2]
  - Hyperelastic material always requires LDA, see [3]
- Contact may be ideal friction free or can support infinite friction



Contact is supported for all FEM model types:

- 3D solid models
- 2D plane stress, plane strain and axial symmetric models

But still be aware of these limitations if you set up a contact model in Creo Simulate:

- In SDA contact analysis, the 3D simulation model may contain all types of elements, idealizations and features, but contact itself is just supported between volume elements (no support of contact between beams, shells or any other elements!)
- In LDA contact analysis, no p-elements requiring rotations in the element formulation may be in the model at all, so no shells and beams; further no advanced and ground springs, no fasteners. Note: rotations at weighted links and advanced rigid links must stay small, since rotations are treated for those like in SDA

1. Introduction

1.2 Repetition of the contact functionalities implemented until Creo Simulate 2.0

Creo Simulate exclusively uses the penalty method to model contact

- In general, in Simulate contact can be computed due to external forces as well as due to an initial interference fit
- Mathematically, in a static contact analysis simulate solves the matrix equation

  $$\left[K(\vec{u}, \vec{f})\right] \cdot \vec{u} = \vec{f}$$

  where the non-linear stiffness matrix [K] is a function of the force vector f and the displacement vector u
- In the simulation model, between the contact flanks nonlinear springs (invisible for the user) are connected to transfer the loads in case of compression
  Note: These (penalty) springs are often called “gap elements” in other FEM codes!
- The stiffness of these springs is adjusted automatically by the software: Simulate tries to iteratively set the penetration depth by adjusting this stiffness to a small value, so that both local stress and the global load balance are accurately captured
- A penetration depth of zero is mathematically impossible, because then the stiffness of these spring elements would become infinite!
- The default setting for the penetration depth at a contact region is based on 5% of the square root of the contact area (value gained from experience). This value can be controlled by advanced users with help of a config.pro and an engine command line option!

1. Introduction

1.2 Repetition of the contact functionalities implemented until Creo Simulate 2.0

Used Newton-Raphson technique and the “Residual Norm Tolerance” in Simulate

- Before convergence of the underlying nonlinear matrix equation

  $$\left[K(\vec{u},\vec{f})\right]\cdot\vec{u}=\vec{f}$$

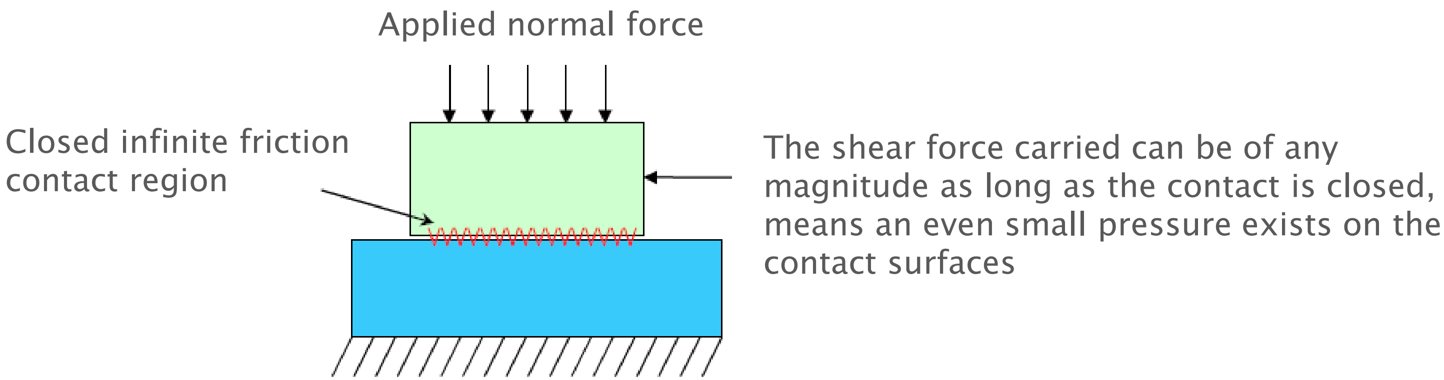
  Simulate calculates the residual error corresponding to the latest solution of the displacement vector u: r=f-Ku. Here, the residual vector r has the dimensions of force (this force must be zero for system convergence). The Newton-Raphson solution then solves for Kdu=r to determine the change in u in the next iteration.
- The residual norm is the dot product r·du. It can be thought of physically as a residual energy, which should be zero when the system has converged. Simulate normalizes the residual norm with the dot product of the total displacement and the total force vector, so the residual norm is: (r·du)/(u·f).
- This residual norm must be smaller than the default value of 1.0E-12 to achieve convergence for the "Residual Norm Tolerance" listed in the engine .pas-file (Note: Until Wildfire 5, the default allowed residual norm was 1.0E-14)
- During this convergence process, the interpenetration depth at the contact flanks is monitored and the spring stiffness may be loosened to improve convergence or tightened (called “adjusted” in the .pas-file) to minimize interpenetration as listed for each iteration process in the engine .pas-file
- For further reading, see [4]

1. Introduction

1.2 Repetition of the contact functionalities implemented until Creo Simulate 2.0

Contact with infinite friction [1]

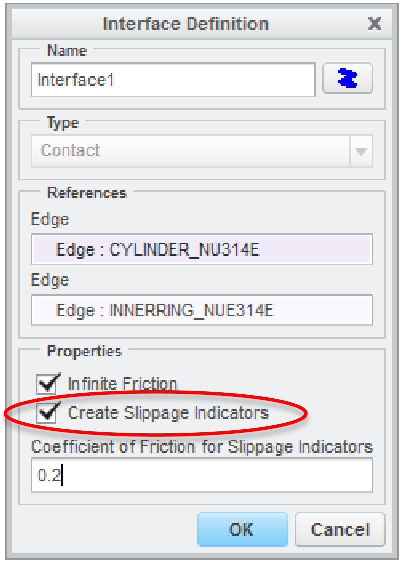
- This is the simplest model for modeling friction contact in a FEM code
- On selection of this model, any large shear load can be accommodated (independent of the magnitude of the pressure load) without sliding occurring:



- After the analysis has run, it is therefore important to check whether the model is still valid or whether under a shear load a slip would occur between the contact surfaces because the friction resistance force (= pressure load x friction coefficient) is too low

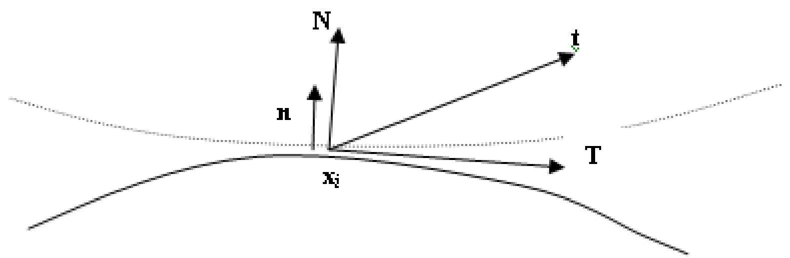
1. Introduction

1.2 Repetition of the contact functionalities implemented until Creo Simulate 2.0



Definition of „Slippage“ in a contact with friction analysis

- Consider an arbitrary point x_i on the edge of the contact with its local normal vector n and the local „Traction Vector“ t:



- The local area based force is now N (with the units of pressure = force/area), the local area based shear force is T („Tangential Traction“). T has the units of shear stress = force/area.
- Slippage at the point x_i does not occur (because of the general law of friction $F_R \leq \mu \cdot F_N$), as long as the locally occurring area-based shear force T is less than the product of area based contact force N and coefficient of friction μ:

$$S_i = T - \mu \cdot N \leq 0$$

- The value of the "slippage" S_i can be seen as being very helpful for checking the validity of the contact analysis: It must be ≤ 0 for a valid model

1. Introduction

1.2 Repetition of the contact functionalities implemented until Creo Simulate 2.0

Measures available in each contact analysis

- *Force* *): Contact force is calculated from the resulting spring force and relative displacements of the gap elements
- *Load*: Contact load is calculated from the integral of the contact pressure (=normal stress) over the contact area (note: this was changed in Creo 3.0, see chapter 1.3!)
- *Area* *): Contact area
- *Maximum contact pressure*
- *Average contact pressure*: Corresponds to the contact *load* divided by the contact *area* (and not measures "contact *force*/contact *area*"!)

*) Default measures automatically created for each user-defined contact

Quality assurance for pressure & stress results at a contact region *until Creo 2.0*:

- If stresses at a contact region are of importance, e.g. the Hertz contact pressure or the max. shear stress below a Hertz contact surface creating pitting, the user should always request the contact load measure in addition to the system-default force measure
- If the mesh is too coarse, until Creo 2.0 the measure contact load typically may give results a magnitude smaller than the contact force, even though both should give identical results
- So the users could easily detect if the engine has underestimated the contact stress by looking at this measure and then simply refine the mesh until both measures are identical!

1. Introduction

1.2 Repetition of the contact functionalities implemented until Creo Simulate 2.0

Additional measures available only in a friction contact analysis

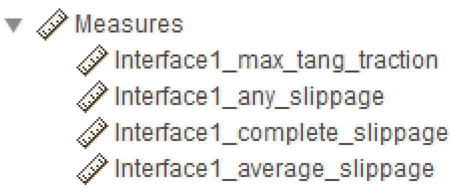
- The „Slippage“ S_i is in general unevenly distributed over the contact area, therefore its characteristic values are made available in the form of three different measurements. Simulate automatically puts these in the engine .rpt-file for true friction contacts, as long as an actual coefficient of friction is specified in the UI:

➢ *InterfaceName_any_slippage*:
better read as „maximum slippage $S_{i,\,max}$ found in the contact region“

➢ *InterfaceName_complete_slippage*:
better read as „minimum slippage $S_{i,\,min}$ found in the contact region“

➢ *InterfaceName_average_slippage*:
Average slippage $S_{i,\,av}$ at the contact region (should be <0 for a valid model)

➢ Additionally computed and put out:
InterfaceName_max_tang_traction:
better read as „maximum contact shear stress in the contact region“



- The characteristic values for the „Slippage“ and the „Tang Traction“ can be found not only as measure in the .rpt-file, but also their complete distribution over the entire contact surface can be seen in the post-processor results
- For the slippage S_i, this is unfortunately limited to red/green plots only (red = $S_i > 0$: invalid model; green = $S_i < 0$: valid contact model; grey: contact surface regions not in contact)

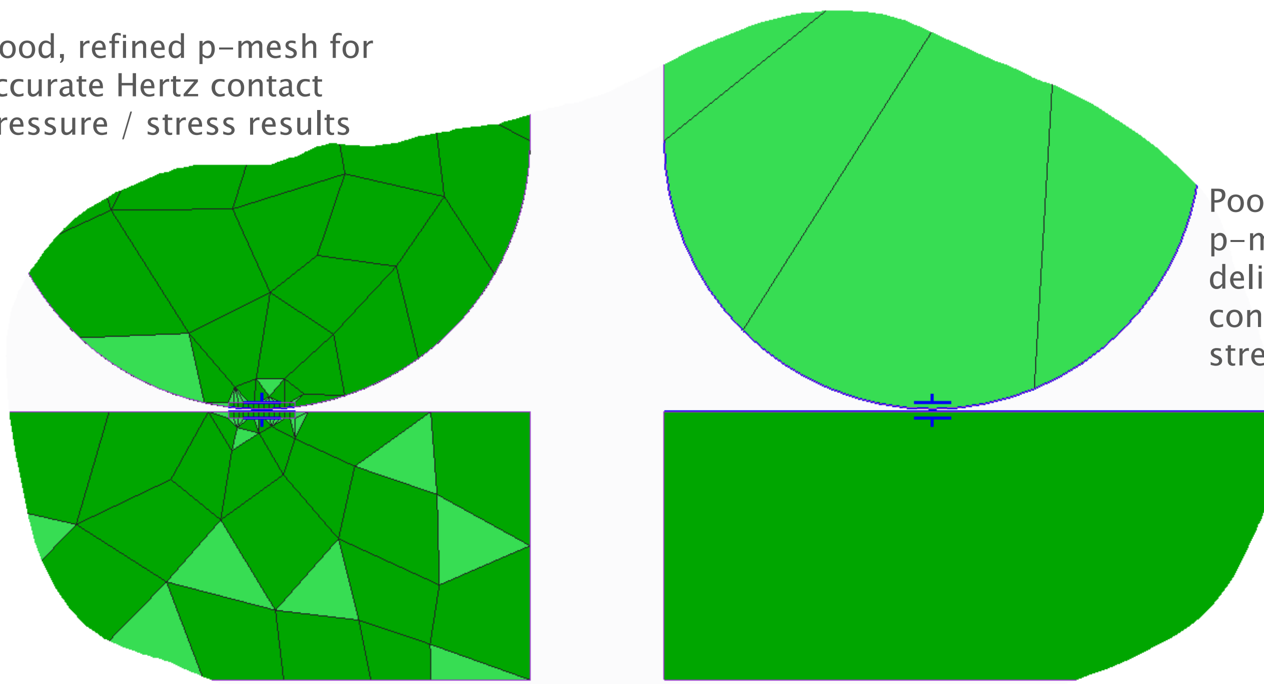
1. Introduction

1.3 Quality assurance of contact pressure & stress results in Creo 3.0

New problem in Creo 3.0:

- In Creo 3.0, unfortunately the contact load measure definition was changed: It is now computed with help of the contact force springs, too, and not any longer by integrating the element normal stress over the contact surface!
- As consequence, the previously described quality check becomes impossible, since the contact load measure will now deliver an identical result to the contact force measure in normal direction, even if the mesh is much too coarse for good stress results:

Good, refined p-mesh for accurate Hertz contact pressure / stress results



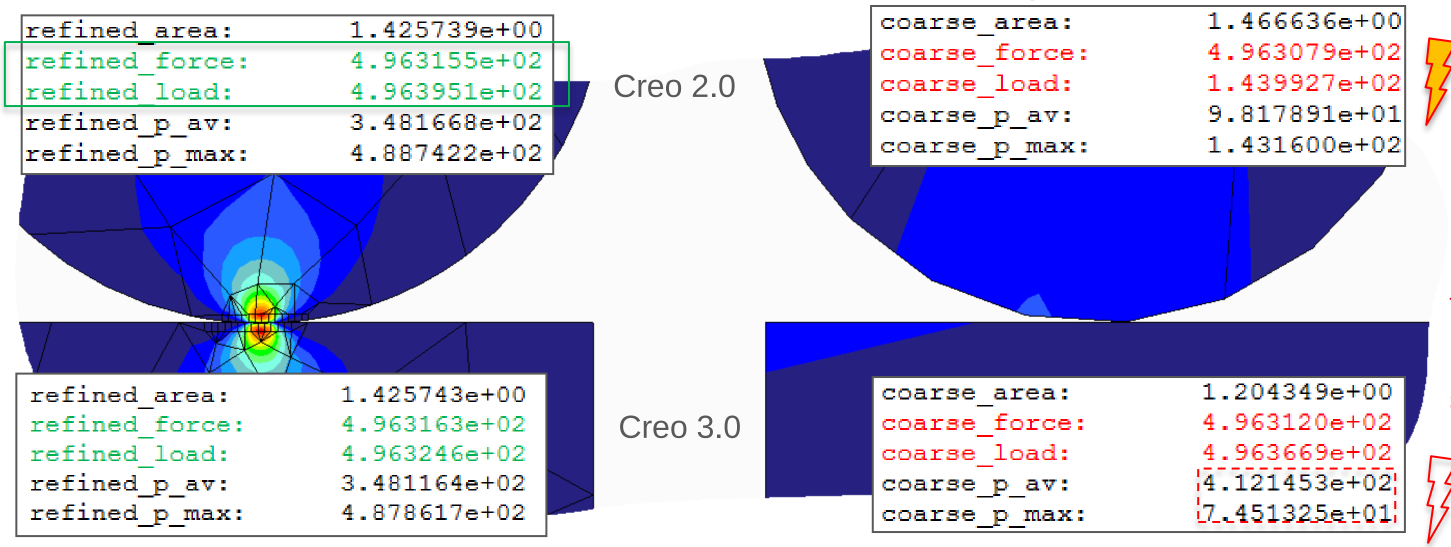
Poor, coarse (default) p-mesh that will deliver incorrect Hertz contact pressure and stress results

1. Introduction

1.3 Quality assurance of contact pressure & stress results in Creo 3.0

- The von Mises stress results and measures of this example become:

force = load measure only for correct results



Different results, the user can immediately detect the problem!

force = load measure even if incorrect stress results!

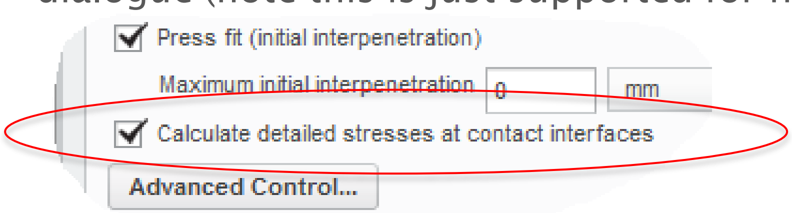
- The author did not find a satisfying alternative criteria to check results, so the user must perform a series of consecutive analyses with refined meshes at the contact regions, respectively, to prove that the contact stress/pressure has converged
- The only error indicator now is that the maximum contact pressure may become smaller than the average contact pressure, but this is by far not as accurate
- The "% convergence-message" for the measure "contact load" in a study performed in multi-pass adaptive convergence is unfortunately NOT sufficient: Contact stress and Hertz contact pressure in a contact analysis can converge to wrong values if the mesh is too coarse (factors too low, see above)!

2. The new contact model with finite friction

2.1 Theory basics

Contact modeling

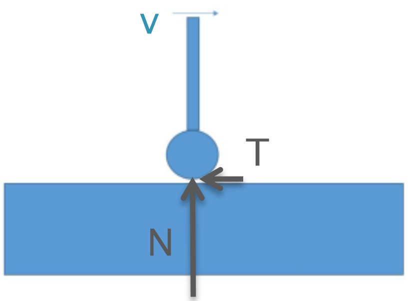
- Simulate 3.0 provides finite sliding interaction between deformable bodies
- It uses a generalized approach of contact between quadratic elements, like described e.g. in detail in [4] (Abaqus 6.12 theory manual)
- Typically, in an h-code, it is checked if a node on one surface contacts a single element face on another surface. A large list of nodes on the one side (the “slave” or “dependent” surface) must be compared against a large list of nodes on the other surface (the “master” or “independent” surface)
- In Simulate, an algorithm is implemented to determine where a point on one surface contacts a point on another surface, where exact geometry is used for both surfaces
- Therefore, since the p-elements can be much larger than h-elements, the element face on the dependent side is sampled at a number of points to see whether it interpenetrates the independent surface
- The user can reduce this number of sampling points to increase speed by unchecking the box “Calculate detailed stresses at contact interfaces” in the analysis definition dialogue (note this is just supported for finite friction contacts!)



Note: Contact between shells or beams is not supported, just between 3D or 2D volumes!

2. The new contact model with finite friction

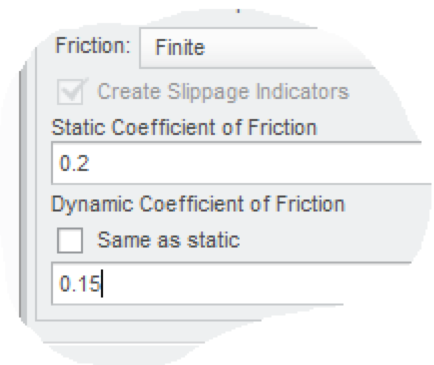
2.1 Theory basics



Penalty method used in finite friction contact

- When the areas in contact are determined, penalty springs are used to prevent interpenetration
- Tangential springs are used if the tangential force does not exceed the force transferrable by static friction
- At locations where the lateral force exceeds the force transferrable by static friction, sliding is permitted, and a tangential traction $T = \mu_{dyn} \cdot N$ is applied, with N=normal pressure and μ_{dyn}=sliding or dynamic coefficient of friction

Stick-Slip



- The algorithm therefore separates between static and dynamic coefficient of friction: The static coefficient is used where sliding does not yet appear, and the dynamic coefficient is used where sliding already appears
- The algorithm reports in the engine files when sliding first occurs at any contact interface using finite friction
- Also for finite friction interfaces, the engine computes the slippage indicators
- Since finite friction contact is just implemented in static analysis, dynamic (inertial) effects are not taken into account
- Anyway, effects from elastic energy stored in the model are taken into account: This (spring) energy is suddenly released if the tangential force exceeds the force that can be transferred by static friction, so we have a simplification of real physics

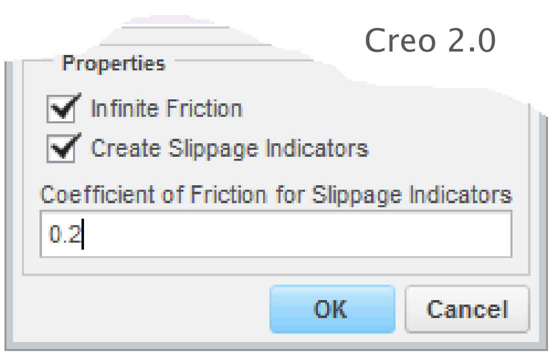
2. The new contact model with finite friction

2.2 New UI functionality

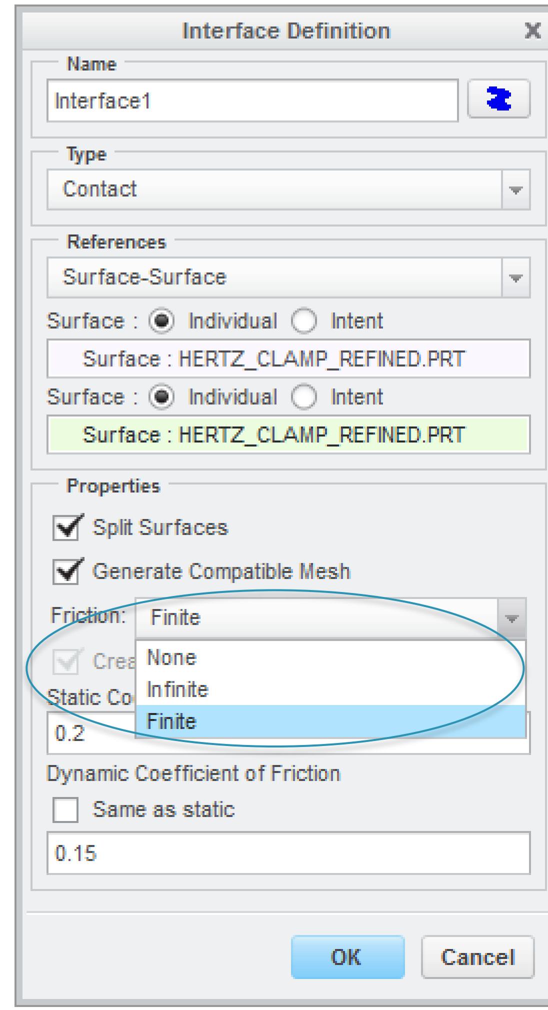
Interface definition

- The interface definition dialogue has been slightly modified compared to Creo 2.0 and does allow to define three contact subtypes for taking into account friction: None, Infinite and Finite
- For infinite friction, creation of slippage indicators is optional if a static friction coefficient is defined; for finite friction, slippage indicators are always created
- For finite friction, the “dynamic” (better sliding) coefficient of friction must be equal (“same as static”) or smaller than the static coefficient of friction

Creo 2.0



Creo 3.0



altran

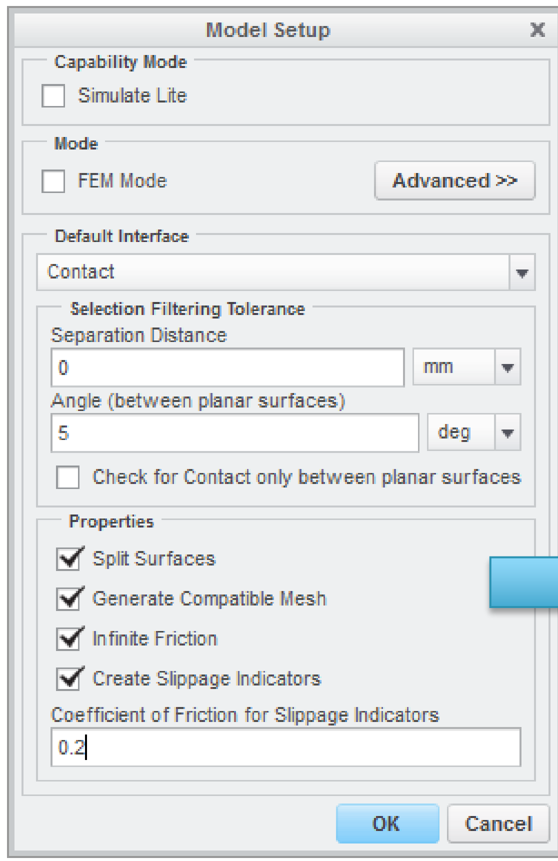
2. The new contact model with finite friction

2.2 New UI functionality

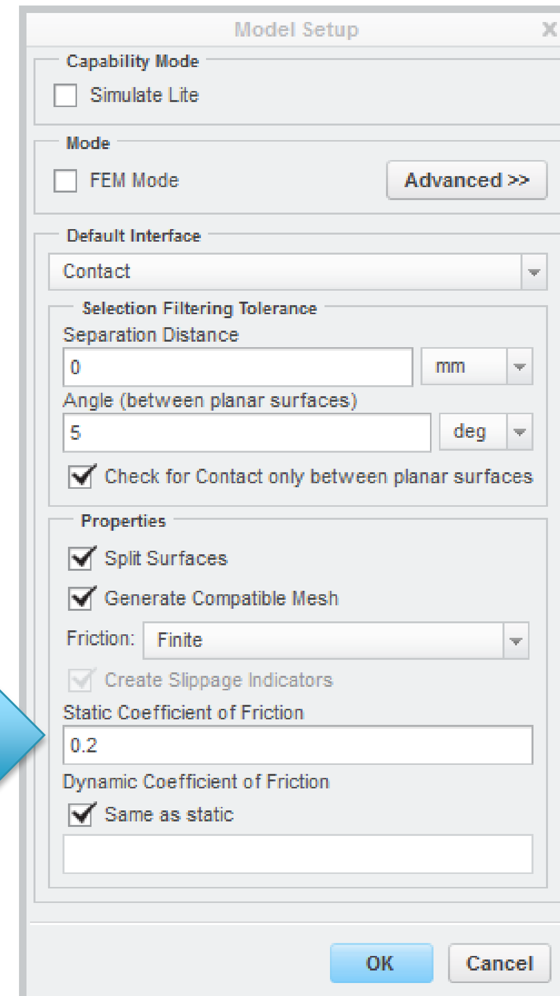
Default Interface definition

- Analog changes have been implemented into the default interface definition in the Simulation Model Setup dialogue

Creo 2.0



Creo 3.0

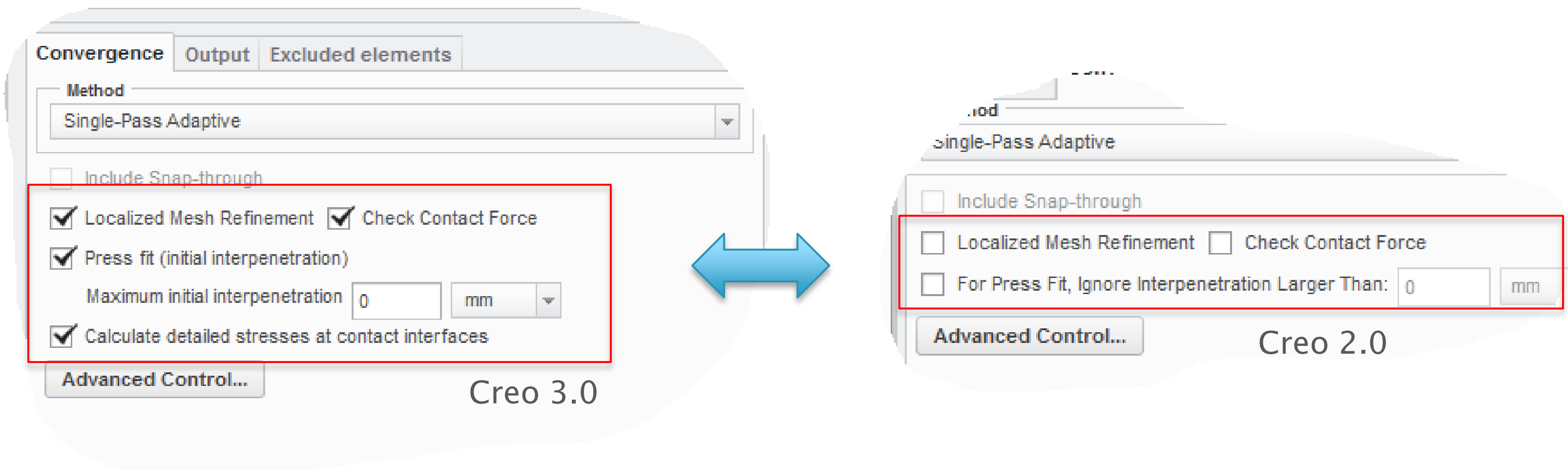


2. The new contact model with finite friction

2.3 Contact analysis definition options

Special Settings for Contact Analysis (1)

- The static analysis definition dialogue in Creo 3.0 now offers 4 options especially for nonlinear contact analysis:



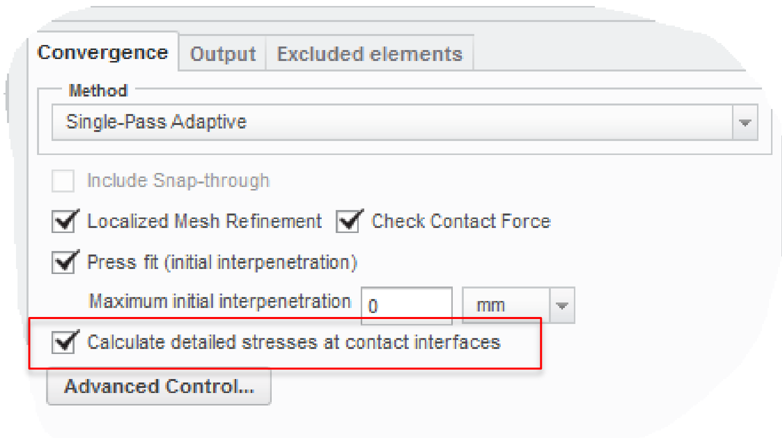
- The explanations given on the following slides are done with the best information available, but own “reverse engineering” tests could not always clearly show the practical influence of these settings
- PTC R&D should provide more detailed information about what these options invoke internally in detail – the online documentation is insufficient here!

2. The new contact model with finite friction

2.3 Contact analysis definition options

Special Settings for Contact Analysis (2)

"*Calculate detailed stresses at contact interfaces*":



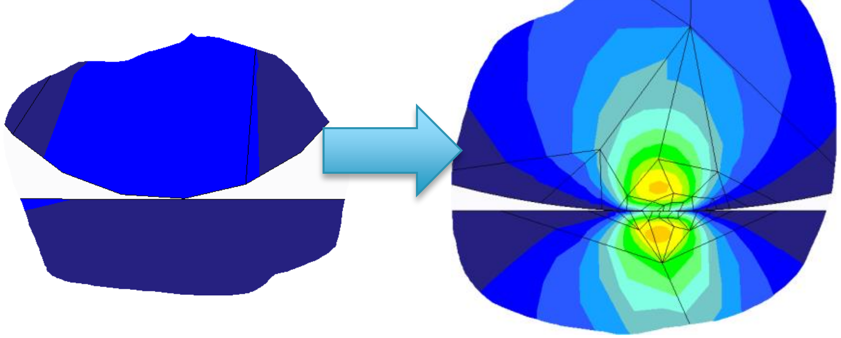
- "*Calculate detailed stresses at contact interfaces*" is new in Creo 3.0 and just can be assessed in LDA analysis of 3D models only if a finite friction contact is in the model – it is not supported for friction-free and infinite friction contacts
- If it is unchecked, the engine uses a reduced amount of sample points in the algorithm that detects finite friction contact between p-element faces, which speeds up the analysis but reduces accuracy in stress results.
- If accurate contact pressures and stresses are of interest, like in Hertz' contacts, this box should always be checked! It should be unchecked only if just the force transfer at the contact is of interest, not the detailed contact stress!

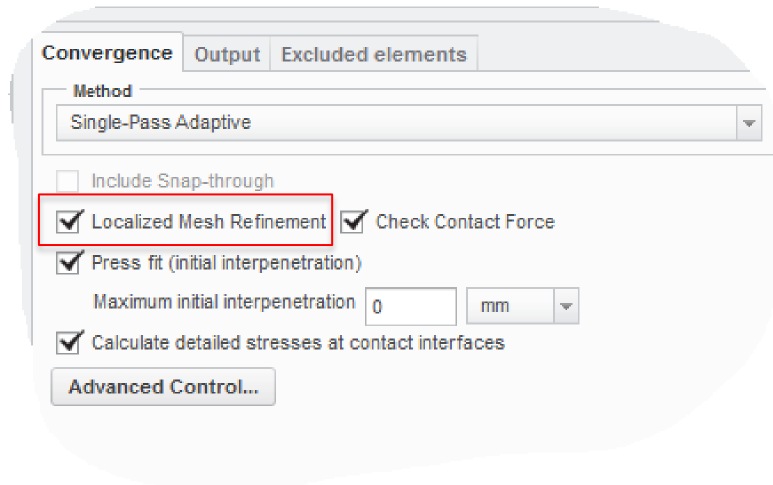
2. The new contact model with finite friction

2.3 Contact analysis definition options

Special Settings for Contact Analysis (3)

"*Localized Mesh Refinement*":

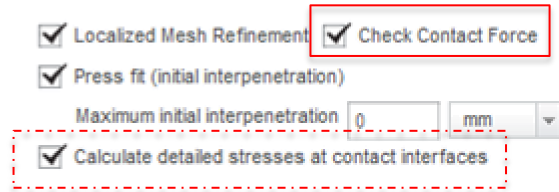




- This causes the engine to request a refined mesh in regions where it is sensed that the contact area is only covering a small part of an element face, leading to inaccuracy
- This checkbox should be activated only if accurate contact pressures are an important objective for the analysis, and if the user did <u>not</u> assure a fine mesh by self-defined mesh controls in the contact region (ideally a mapped mesh with undistorted brick elements, which undoubtful delivers better results compared to tetrahedron meshes of the automatic refinement function)
- If the mesh refinement fails during the first pass, Simulate continues with a second pass using the original mesh.
- During the second pass, the user can review the results of the first pass (e.g. contact pressure distribution) in the postprocessor and stop the analysis if the results are not satisfactory

2. The new contact model with finite friction

2.3 Contact analysis definition options



Special Settings for Contact Analysis (4)

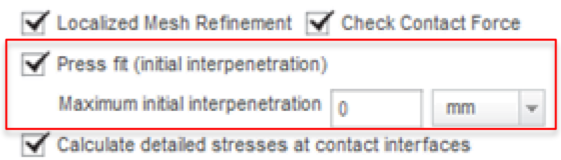
"*Check Contact Force*":

- Acc. to actual PTC R&D information, the checkbox "Check Contact Force" invokes the engine to tighten the contact spring if the contact force – calculated from the contact spring force and *not* the surface normal stresses – changes more than 5 % compared to the previous iteration. The contact force is said to be converged, if that change is less than 5 % in consecutive iterations. The value of 5 % is "hard wired" and cannot be influenced by other user defined convergence accuracy settings in the analysis definition dialogue (valid for Creo 2.0 and Creo 3.0)
- In Creo 3.0 (P20), this extra check for force convergence in addition to the default convergence checks will be applied only if user also chooses the option "Calculate detailed contact stresses at contact interfaces". Activating the option is not necessary unless the user wants detailed contact stresses
- According to older R&D information, this checkbox causes the engine to compare the force from springs to the force from normal stresses at a contact interface
- "*Check Contact Force*" should therefore only be activated if a very flexible model is present where the default interpenetration test is allowing too much interpenetration. Turning on "check contact force" makes the contact springs become very stiff
- If the contact region shows singular stresses, like existing at the boundary of a stiff planar surface touching a bigger stiff planar surface, the stress-based force is inaccurate. The engine keeps tightening the springs trying to get the two to agree, which may cause problems, see Case 11273040/SPR 2171011

altran

2. The new contact model with finite friction

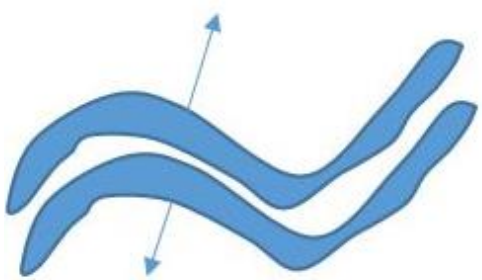
2.3 Contact analysis definition options



Special Settings for Contact Analysis (5)

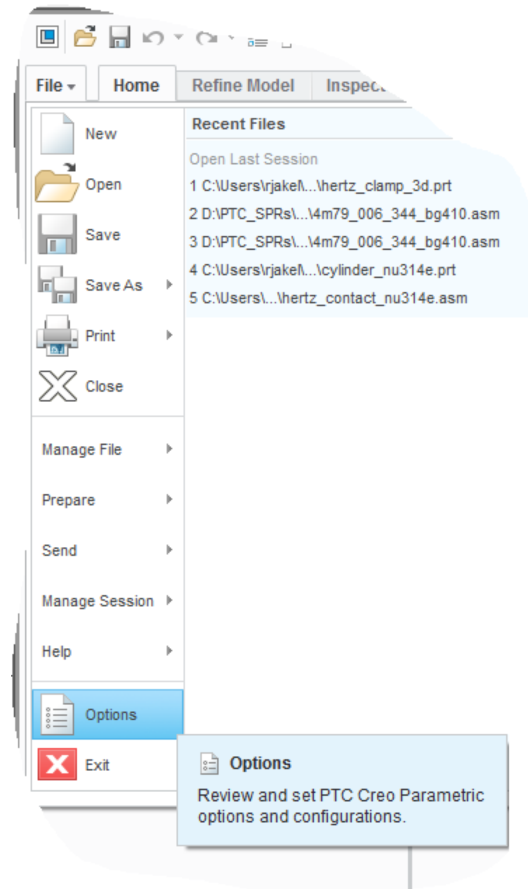
"*Press fit (initial interpenetration)*":

- In earlier (pre-Creo) releases, Mechanica always used to automatically sense pressfit in load step 0 by checking interpenetration using the undeformed geometry
- But since methods have been developed allowing to create large numbers of contacts automatically, the code can sometimes incorrectly think there is interpenetration
- An example is shown in the figure below: The user could request that contact will be checked between all the surfaces of the upper and lower bodies
- Interpenetration is then sensed when two surfaces have opposite normals and the distance vector between the independent and dependent side has a negative dot product with the normal to the independent side. As shown in the figure, this could incorrectly be detected for the top surface of the top body compared to the bottom surface of the bottom body
- Therefore, *checking for initial interference is now only done if the user checks the box!*
- By knowing the entered maximum expected interference, the engine will ignore a detected interpenetration if it exceeds the user's maximum expected value



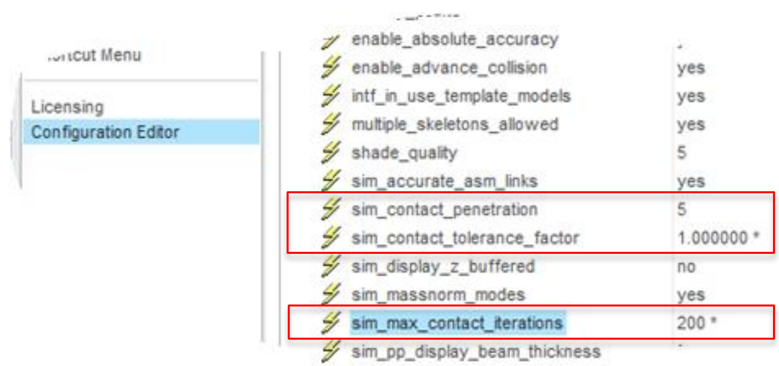
2. The new contact model with finite friction

2.4 Additional config.pro and engine command line options for contact analysis



Motivation

- Nonlinear (contact) analyses naturally do not run as stable as linear analyses
- Even though in Simulate some effort was spent to make the nonlinear algorithms robust without user interaction, it happens that contact analyses do not converge or give inaccurate or even wrong results, often without any warning
- Therefore, a couple of (unfortunately well hidden) additional options are coded that allow experienced users to influence the solution process
- In pre-Creo releases, only engine command line options or environment variables were available for this purpose, as described in [1]
- Since Creo 2.0, now additionally the most important options can be controlled more easily with help of the configuration editor: *File > Options > Configuration Editor*



2. The new contact model with finite friction

2.4 Additional config.pro and engine command line options for contact analysis

Additional options to influence nonlinear contact analysis (1)

- Config.pro-option *"sim_contact_penetration" p*:
  (=engine command line option: *-contact_penetration* p)
  The default penetration depth at a contact is 5% of the square root value of the contact area. Enter p as positive real number between >0 % and 100 % to modify the default value of 5 %
- Decreasing this value tightens the penalty springs at the contacts so that penetration is minimized, but entering values too close to Zero leads to "infinitely stiff" penalty springs making it impossible for the solver to converge
- Usually it does only make sense to decrease this option stepwise e.g. in potencies of 10 (e.g. 0.5 %, 0.05 %, 0.005 %…), see the convergence study in [1]. An increase to values >5 % up to 100 % usually does not help!
- Note: The meaning of this config.pro option as well as the engine command line option was different before Creo 2.0 M100 and Creo 3.0 M020: There, p was the multiplication factor for the max. allowed default penetration depth of 5%. If you set p to 0.01 for example, the maximum penetration depth is reduced to 0.0005 absolute (=0.05% of the square root value of the contact area)!
- Also note that since Creo 3.0 M040, the UI accepts any real number here between 0 and 100 instead of only integers like currently in Creo 2.0 (0%, 1%, 2% … 100 %)
- Never enter a Zero here even though currently the UI does allow you to do so, see case C12858014, otherwise the analysis will fail (e.g. with fatal error "insufficiently constrained")

2. The new contact model with finite friction

2.4 Additional config.pro and engine command line options for contact analysis

Additional options to influence nonlinear contact analysis (2)

- Config.pro-option "*sim_max_contact_iterations*" *n:*
  (= Engine command line option: *-contact_nr_its* n)
  Specifies the maximum allowed number n of iterations for contact analysis. The maximum number of iterations is n=200 by default. The iterations will stop if the analysis reaches convergence or the maximum number of iterations is reached in case no convergence has been obtained
- Config.pro-option "*sim_contact_tolerance_factor*" y:
  (= Environment variable: MSE_CONTACT_TOLERANCE_FACTOR y)
  Use this option to modify the residual norm tolerance used for contact convergence. This option acts as a multiplication factor y for the residual energy norm, which by default must be below 1.0 e-12 for an analysis to converge. The default value of this option is 1. If for example you set y to 1.0E4, the residual norm tolerance is increased to 1.E-08.

  An appropriate value for y can be determined by examining the residual norm values reported in the engine .pas-file ("Checkpoints"-tab): For example, if these values are approximately 5E-11, then the analysis is failing to reach the default by a factor of about 50. The option should be set to 50 or higher. If with an upper limit as high as 10000 the analysis still does not converge, you may need to check the model itself. Allowing too high residual norms will lead to inaccurate or even wrong results, usually there is no reason to change the default!

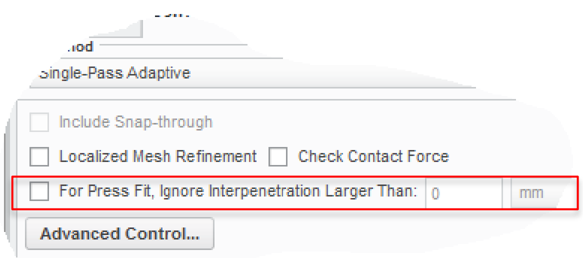
2. The new contact model with finite friction

2.4 Additional config.pro and engine command line options for contact analysis

Additional options to influence nonlinear contact analysis (3)

The following environment variables/engine command line options are not supported as config.pro option and are usually not necessary. You may try them if your contact model does not deliver satisfying results:

- Environment variable "*MSE_CONTACT_LENGTH_CHECK*" (set to e.g. yes or true):
  Acc. to PTC R&D, this ENV is useful specially for SDA contact analysis. If set, the engine aggressively keeps checking interpenetration and tightens contact springs accordingly.
  Remark: In own tests at a problem model with infinite friction, the author could not observe a beneficial effect of this environment variable, but it worsened the situation, see C12900431/SPR4877899
- Environment variable "*MSE_CONTACT_INTERPENETRATION_TOLERANCE*" x:
  This is an environment variable used as workaround for Creo Elements/Pro 5.0 users only, if Mechanica erroneously found interpenetrations e.g. at very thin surfaces in load step 0 even though there are none. Users can set it to 0.0 in this case. For more details, see SPR 1983693 and document CS6933 (09-May-2015).
  This behavior was corrected in Creo 1.0 F000 (see option below).



2. The new contact model with finite friction

2.4 Additional config.pro and engine command line options for contact analysis

Additional options to influence nonlinear contact analysis (4)

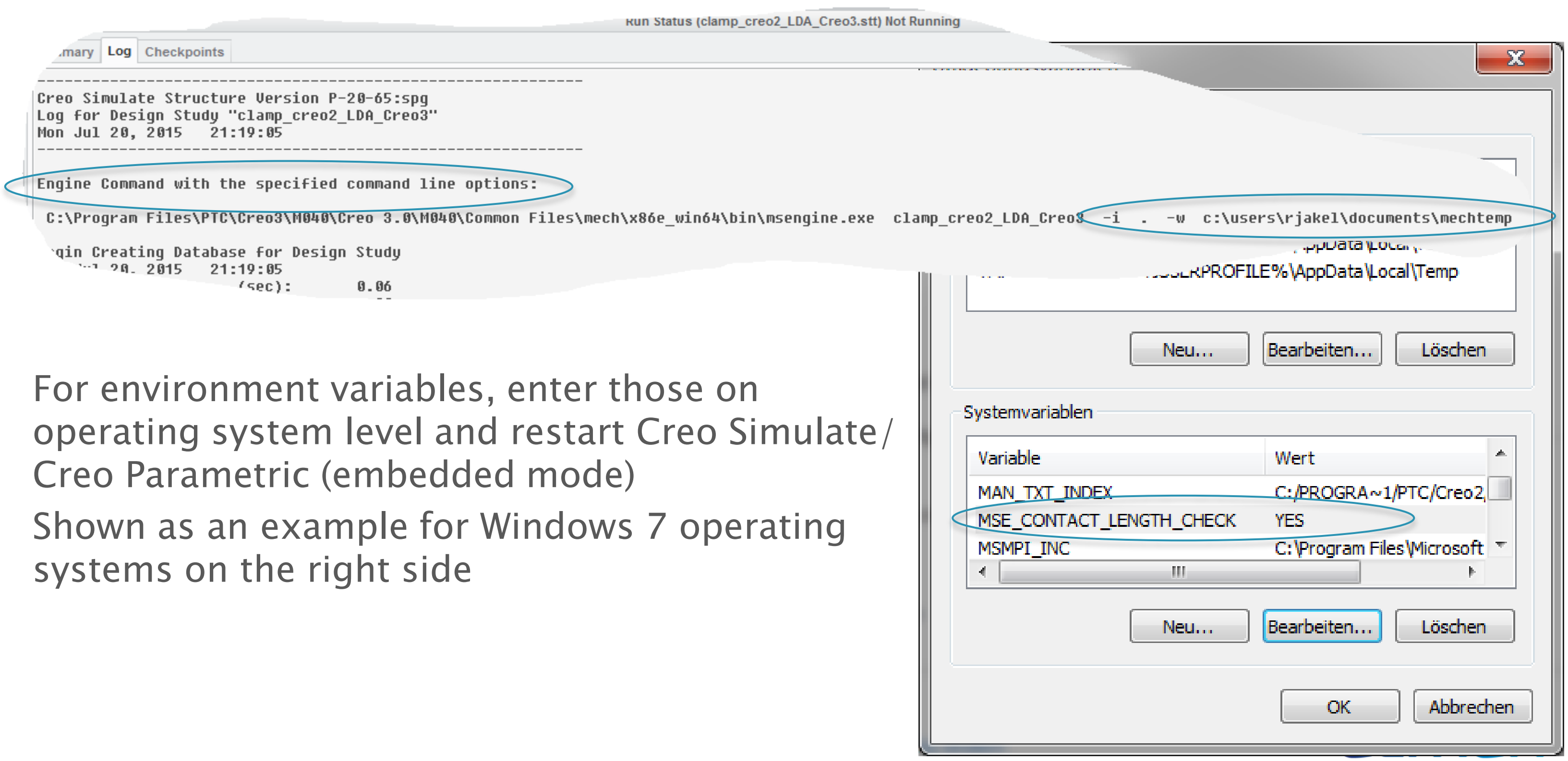
- Engine command line option *"-contactSpringRatio"* y:
  By definition: contactSpringRatio = $K_{cs,ini}$ / $K_{ele,max}$
  with
  $K_{cs,ini}$ = initial contact spring stiffness for a certain region
  $K_{ele,max}$ = maximum element stiffness found in that region
- The default value for this ratio is set to y=0.1, so in a contact region we have $K_{cs,ini} = 0.1 * K_{ele,max}$
- However, it is possible for some models that the initial estimate of the contact spring stiffness may come very low. The typical symptoms include:
  1. Convergence achieved without adjusting springs – no spring adjustment messages in the .pas-file;
  2. only spring adjustment/tightening messages and no spring loosening messages in the .pas-file and/or
  3. large penetrations in converged solution.
- To allow the user to tune the initial contact spring stiffness in such models correctly, this engine command may be useful. For example, if the user specifies this ratio via engine line command as -contactSpringRatio 100.0, then the initial contact spring stiffness for a region becomes $K_{cs,ini} = 0.1*100.0* K_{ele,max} = 10* K_{ele,max}$ of that region. So, the initial contact springs are now 10 times stiffer than the stiffest element in the corresponding region

2. The new contact model with finite friction

2.4 Additional config.pro and engine command line options for contact analysis

Some hints for using engine command line options and environment variables:

- Start the Simulate analysis in batch mode
- Write the command line option(s) with help of a text editor into the list of existing command line options of the mecbatch.bat-file
- The engine command line options (all starting with a "-") that were really used during analysis are then reported in the engine .stt-file ("Log" tab):



- For environment variables, enter those on operating system level and restart Creo Simulate/ Creo Parametric (embedded mode)
- Shown as an example for Windows 7 operating systems on the right side

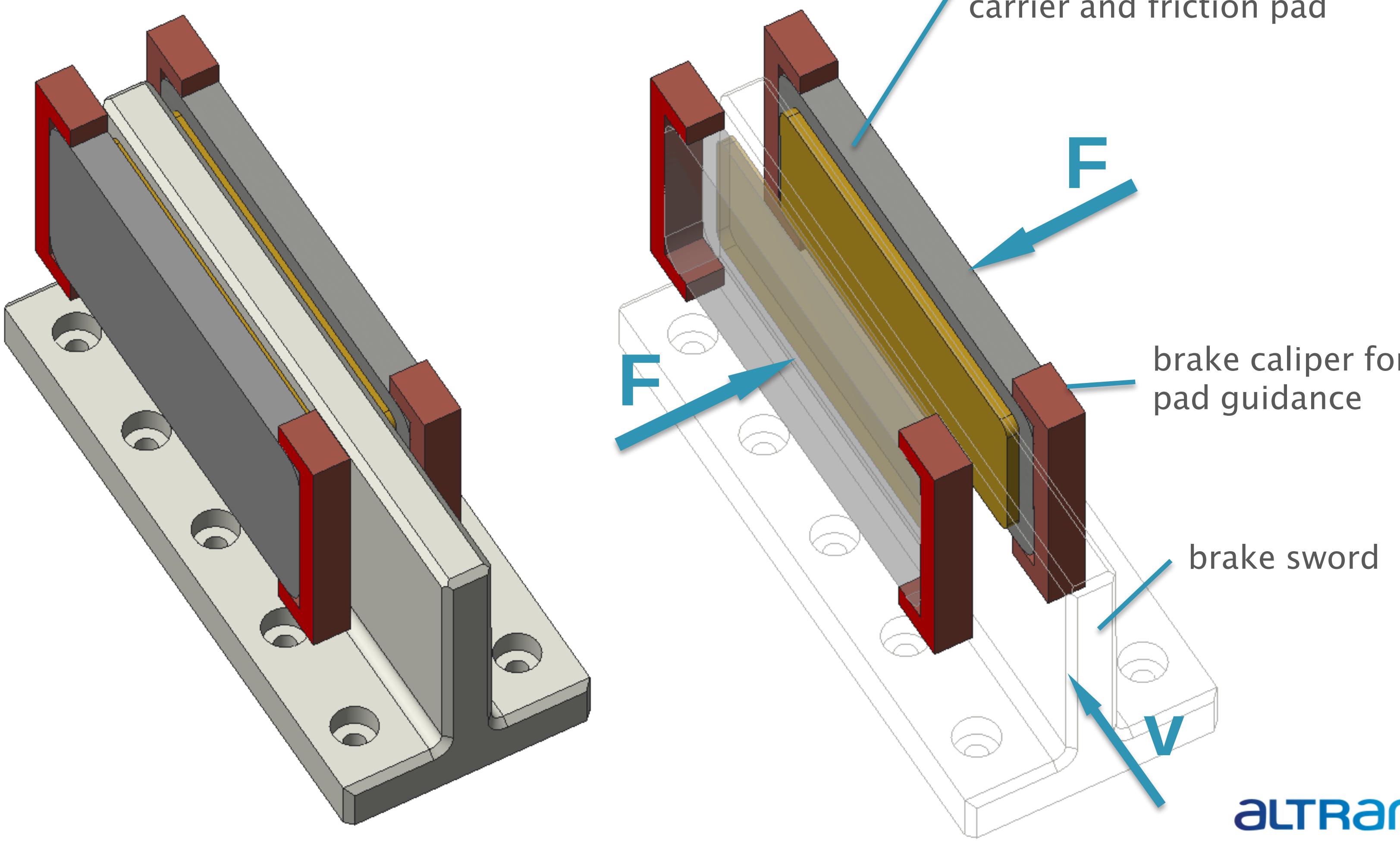
Part B: Application Examples

1. Brake system with brake pad and brake sword

1.1 Model description

The example CAD model

- Two brake pads clamp a brake sword

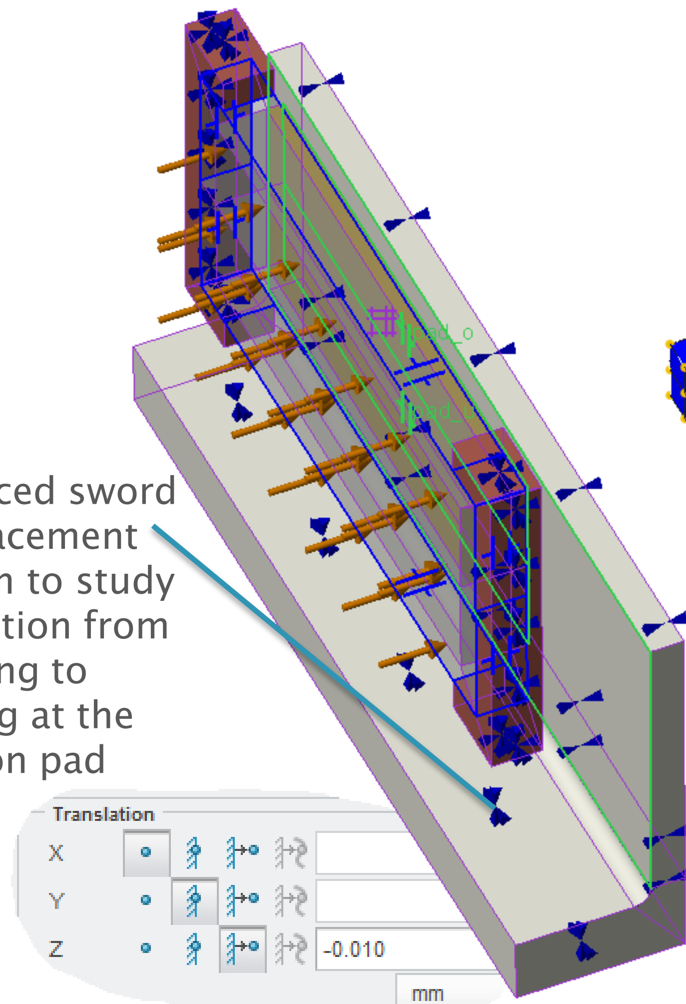


1. Brake system with brake pad and brake sword

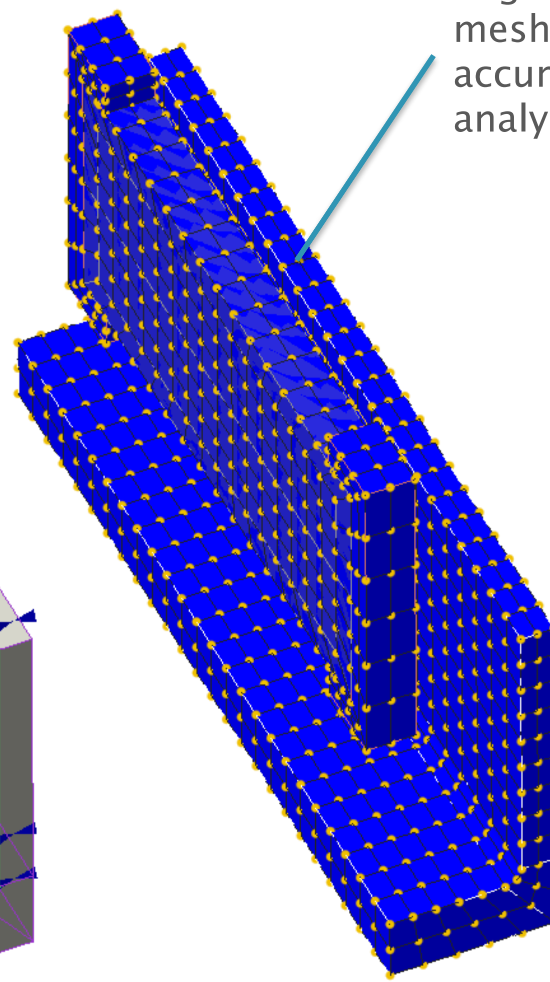
1.1 Model description

Simplified Simulation model

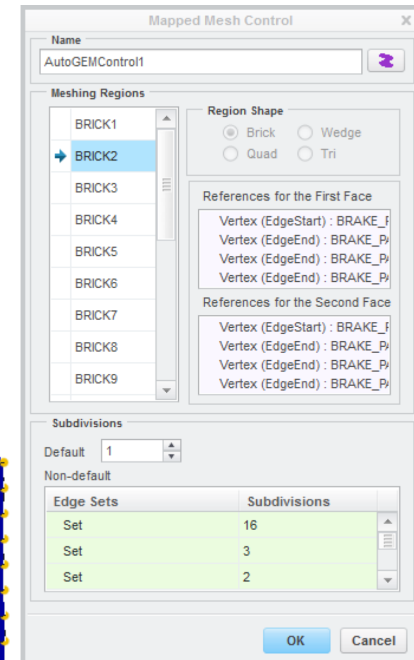
- Half model with mirror symmetry



Enforced sword displacement 10 µm to study transition from sticking to sliding at the friction pad



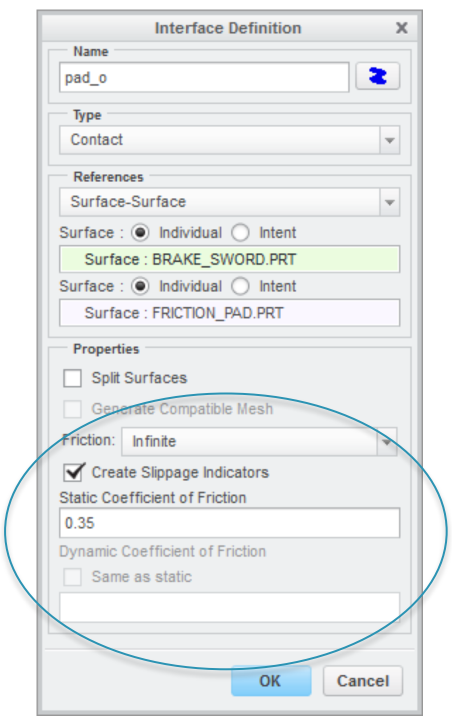
Regular mapped p-brick mesh for increased accuracy and minimized analysis time

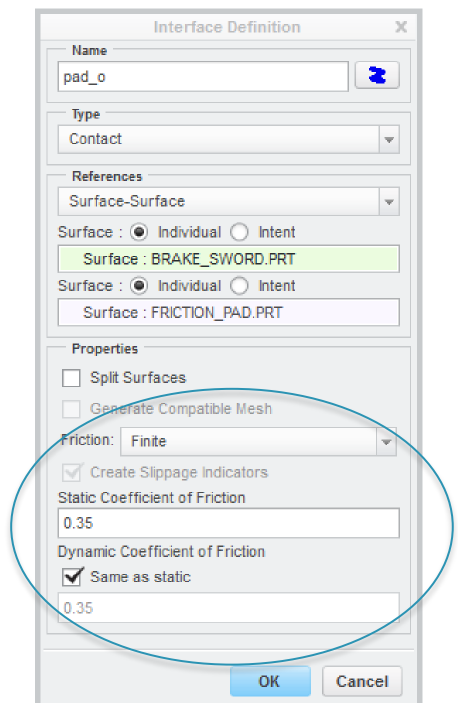


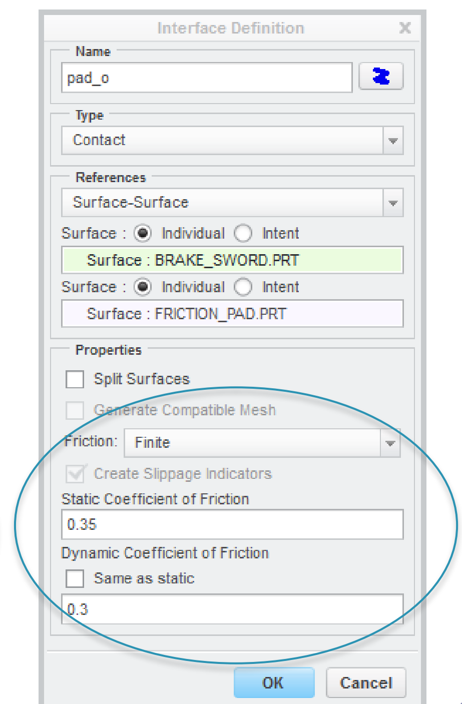
1. Brake system with brake pad and brake sword

1.2 Technical data and friction definition

- Clamping force 9600 N (≈10 kN) (reflects friction pad nominal unit pressure = 1 MPa)
- All contacts brake pad – brake caliper are ideal friction free for simplicity
- Three different contact definitions between friction pad and brake sword for software testing and understanding:
  - Infinite friction contact with μ=0,35 (just for slippage indicator calculation)
  - Finite friction with $\mu_{static}=\mu_{dynamic}=0,35$
  - Finite friction with $\mu_{static}=0,35$ and $\mu_{dynamic}=0,3$





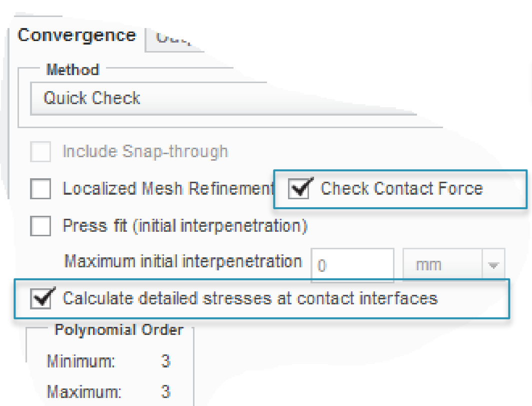


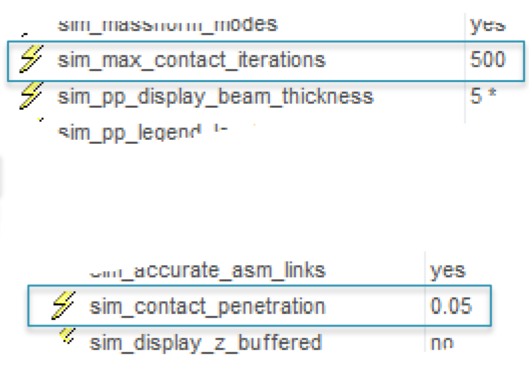
1. Brake system with brake pad and brake sword

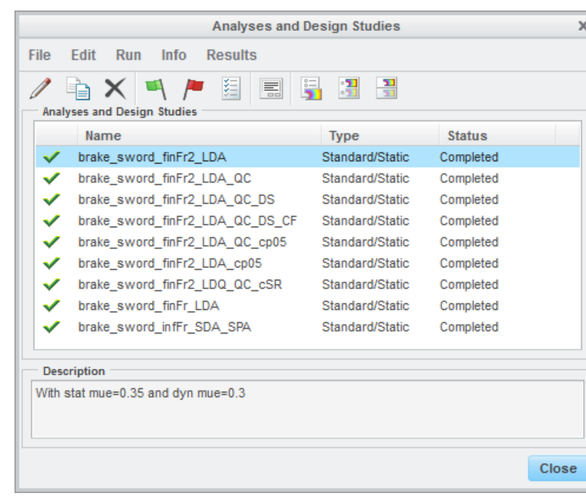
1.3 Performed analyses

A bunch of different analyses was performed to test & understand software behavior:

- SPA (Single pass Adaptive with 5 % RMS stress error) analysis with the known infinite friction model as SDA (Small Displacement Analysis) for reference purposes
- SPA analysis with the new finite friction model and $\mu_{static}=\mu_{dynamic}$ (LDA)
- SPA analysis with the new finite friction model and $\mu_{static}=0{,}35$ and $\mu_{dynamic}=0{,}3$
  - ➢ With default settings
  - ➢ With contact penetration = 0,05 %
- A couple of additional analyses with $\mu_{static}=0{,}35$ and $\mu_{dynamic}=0{,}3$ in quick check convergence only and using
  - ➢ Default settings
  - ➢ Activated detailed stresses at contact interfaces
  - ➢ Activated detailed stresses at contact interfaces and check contact force
  - ➢ Default settings and contact penetration = 0.05 %





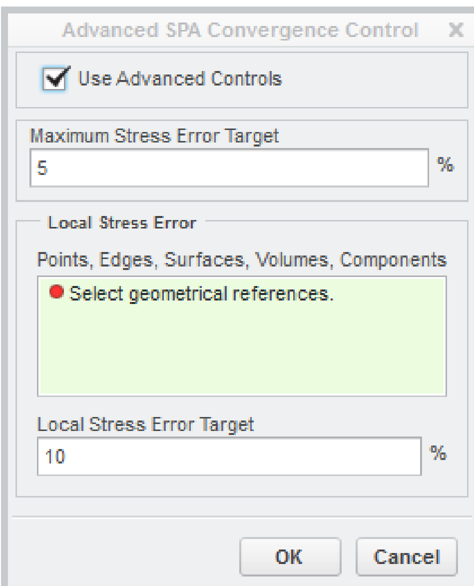


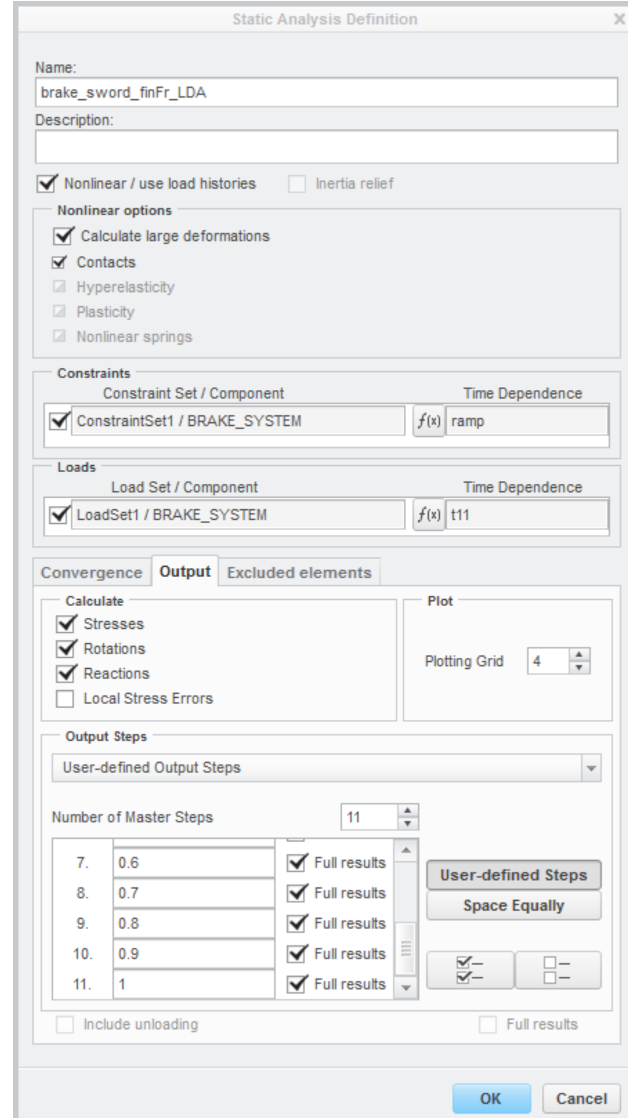
1. Brake system with brake pad and brake sword

1.3 Performed analyses

Note for all analyses:

- Full result output is requested for each µm enforced displacement
- All SPA analyses have been computed with 5 % instead of default 8 % local RMS stress error target and the shown fine regular brick mesh, so the user should usually expect very good numerical results!



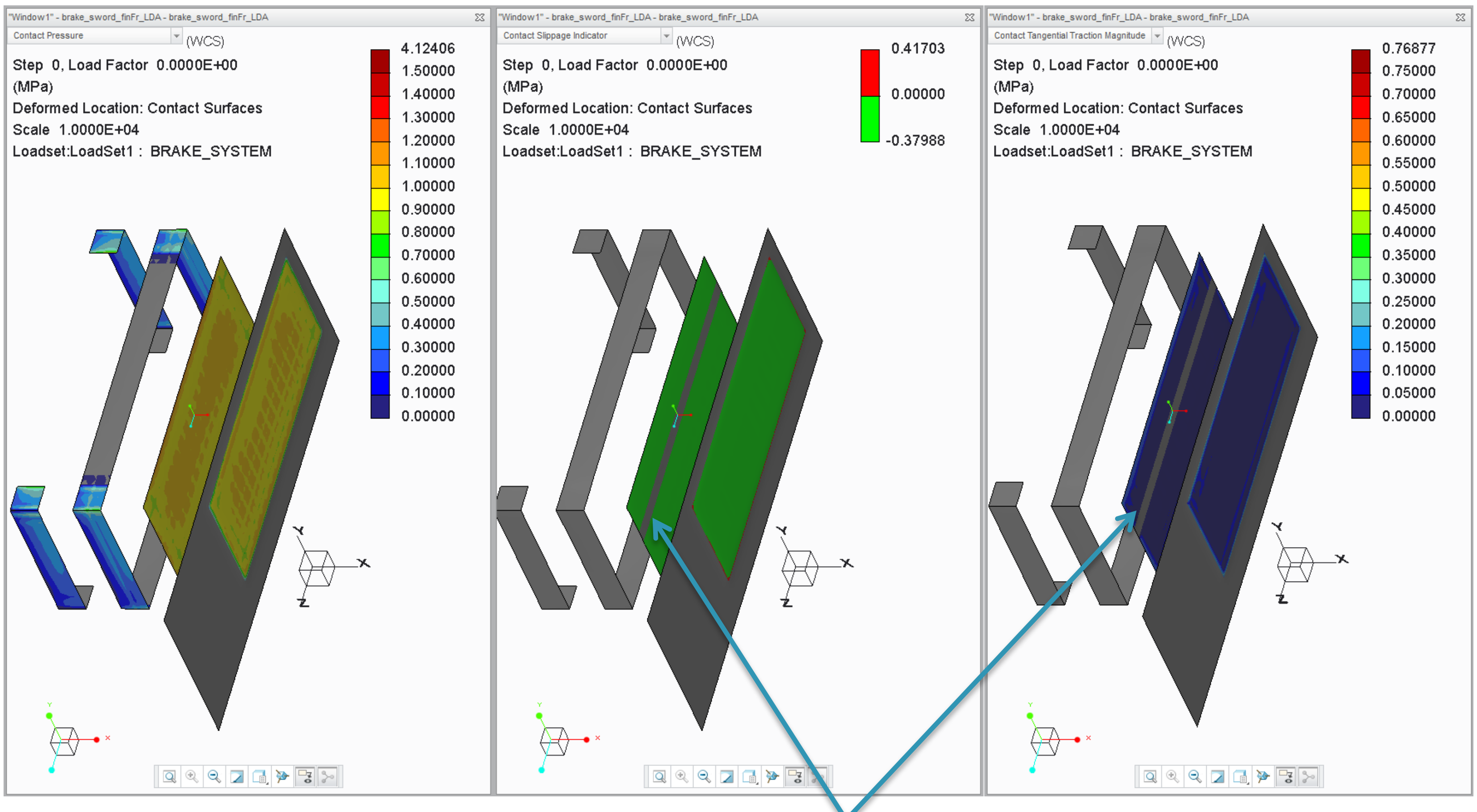


1. Brake system with brake pad and brake sword

1.4 Results | finite friction model | $\mu_{static}=\mu_{dynamic}=0.35$

Load step 0: Only clamping force 9,6 kN = Pressure 1 MPa applied (disp scale 10000)

- Contact pressure [MPa]
- Contact slippage indicator [-]
- Contact tangential traction [MPa]



Exploded view!

Bug – stripe with no results at contact surface in contact

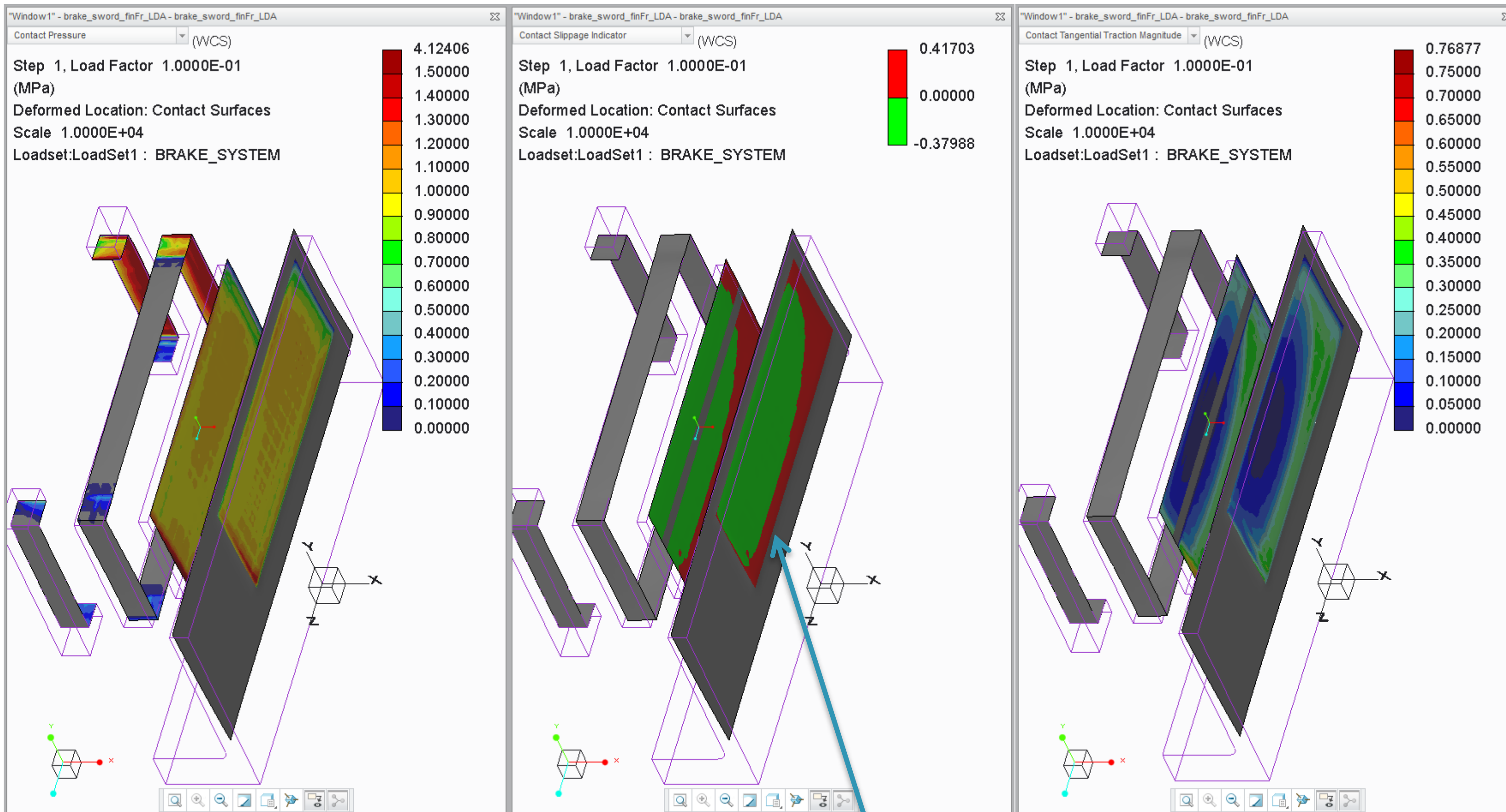
1. Brake system with brake pad and brake sword

1.4 Results | finite friction model | $\mu_{static}=\mu_{dynamic}=0.35$

```
Begin Time Step 1 of 10:  1.00000e-01
Tue Mar 08, 2016    22:18:09
*** sliding first Occurred
```

Load step 1: Clamping force 9,6 kN and 1 µm enforced sword displacement (scale 10000)

- System deforms elastically, some local (see pas- & rpt-file message), but no global sliding



Exploded view!

Local sliding at red regions only

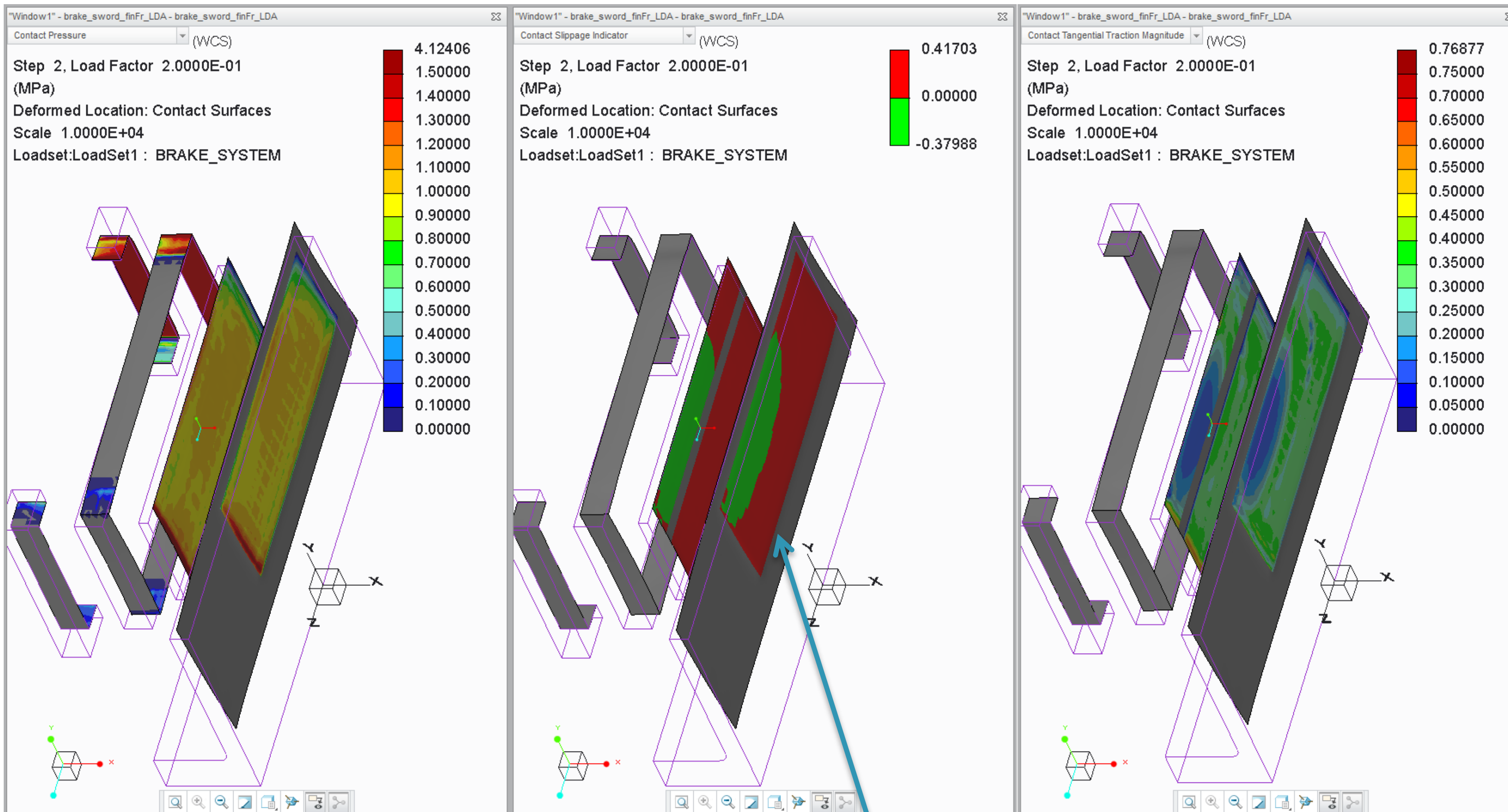
1. Brake system with brake pad and brake sword

1.4 Results | finite friction model | $\mu_{static}=\mu_{dynamic}=0.35$

No fully sliding message in the pas-file!

Load step 2: Clamping force 9,6 kN and 2 µm enforced sword displacement (scale 10000)

- System further deforms elastically, some further local, but still no global sliding

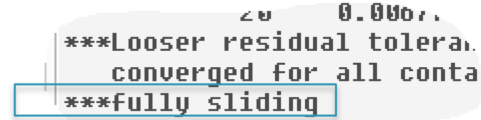


Exploded view!

Local sliding at red regions only

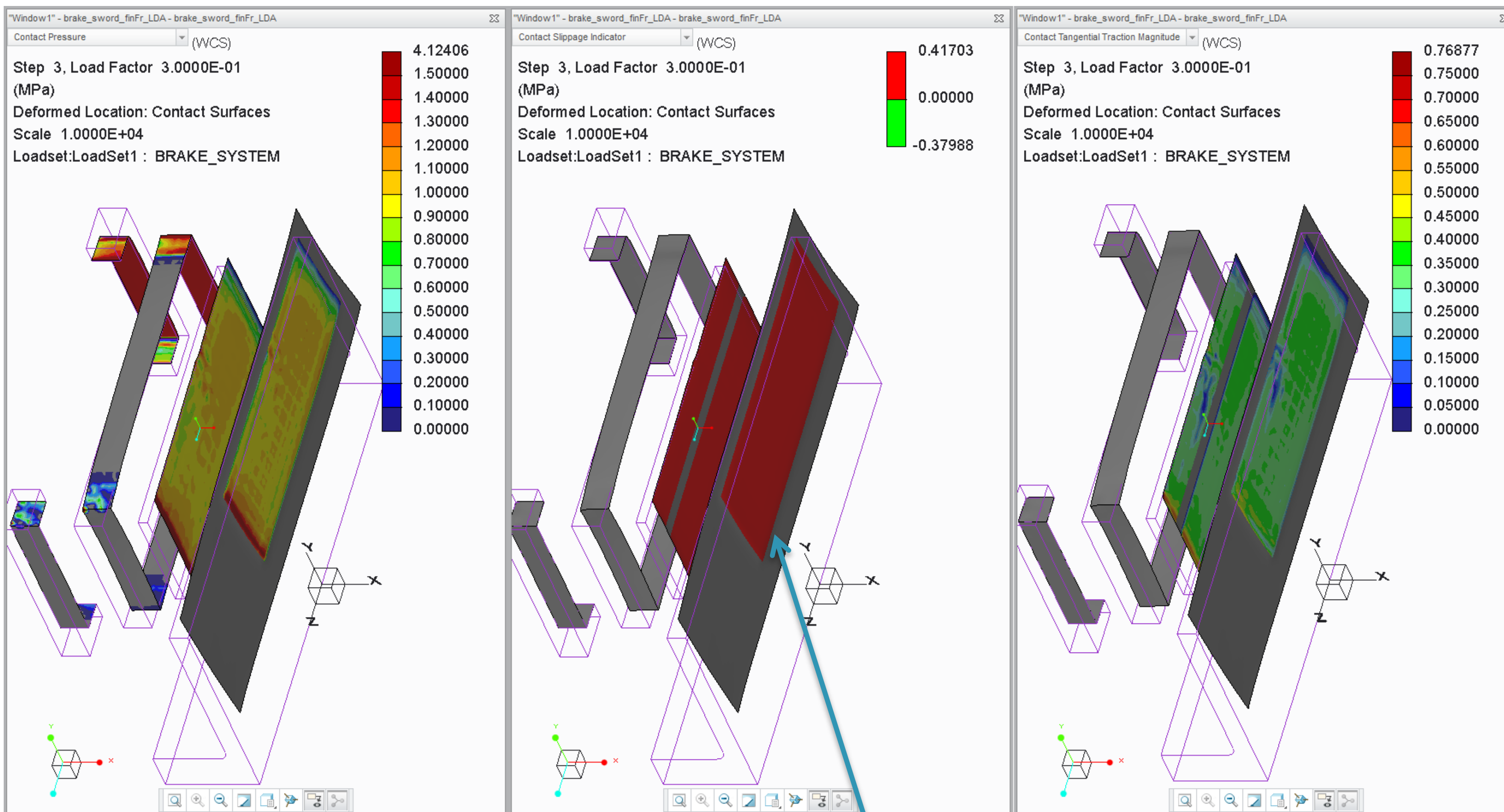
1. Brake system with brake pad and brake sword

1.4 Results | finite friction model | $\mu_{static}=\mu_{dynamic}=0.35$



Load step 3: Clamping force 9,6 kN and 3 µm enforced sword displacement (scale 10000)

- System fully slides (see message in the engine *.pas file)



Exploded view!

Fully sliding, all is red!

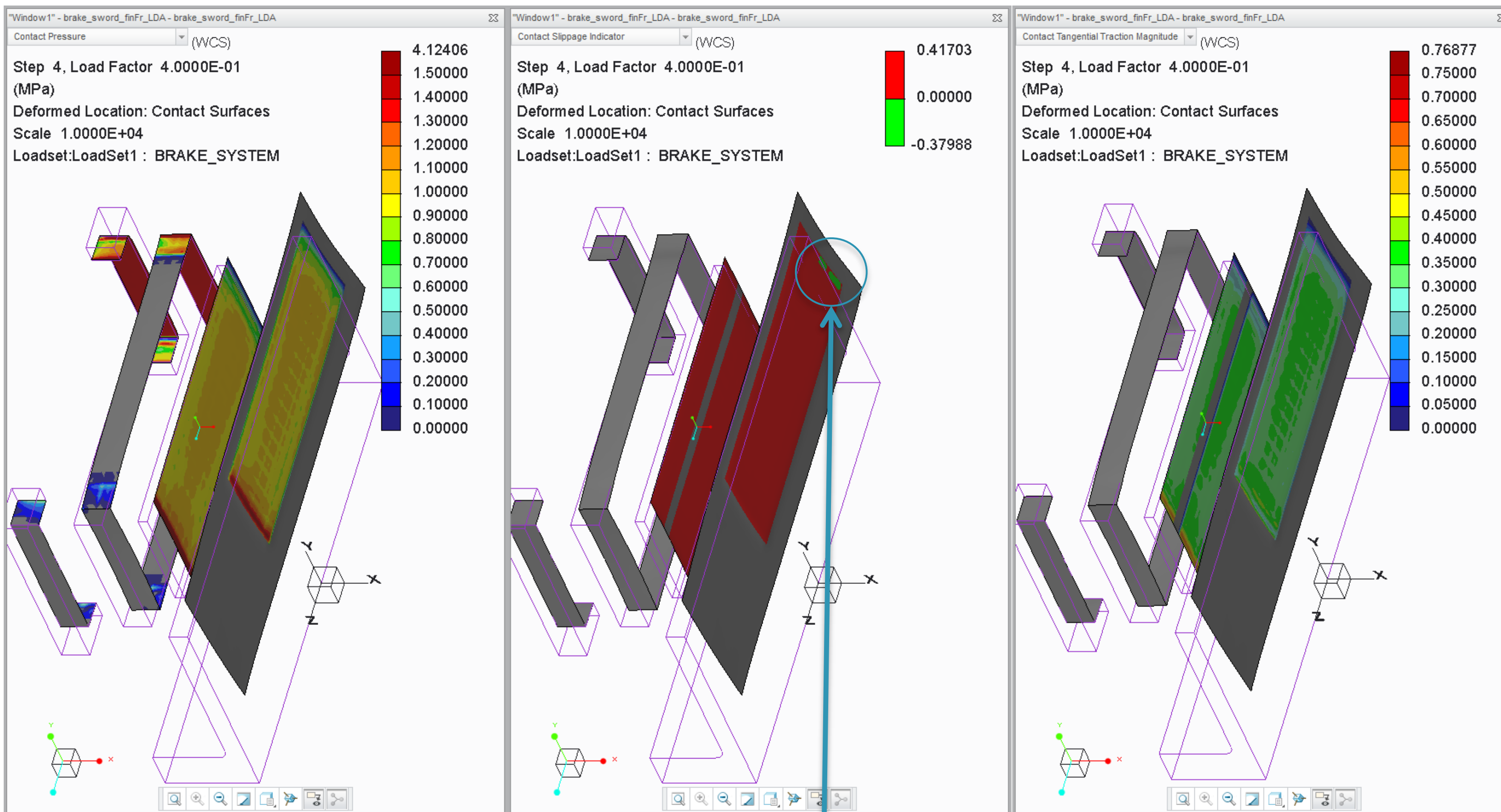
1. Brake system with brake pad and brake sword

1.4 Results | finite friction model | $\mu_{static}=\mu_{dynamic}=0.35$

No fully sliding message in the pas-file!

Load step 4: Clamping force 9,6 kN and 4 µm enforced sword displacement (scale 10000)

- System fully slides (see messages in the engine *.pas file)



Exploded view!

For this system, if $\mu_{static}=\mu_{dynamic}$, after fully sliding appeared, it can not stick again!

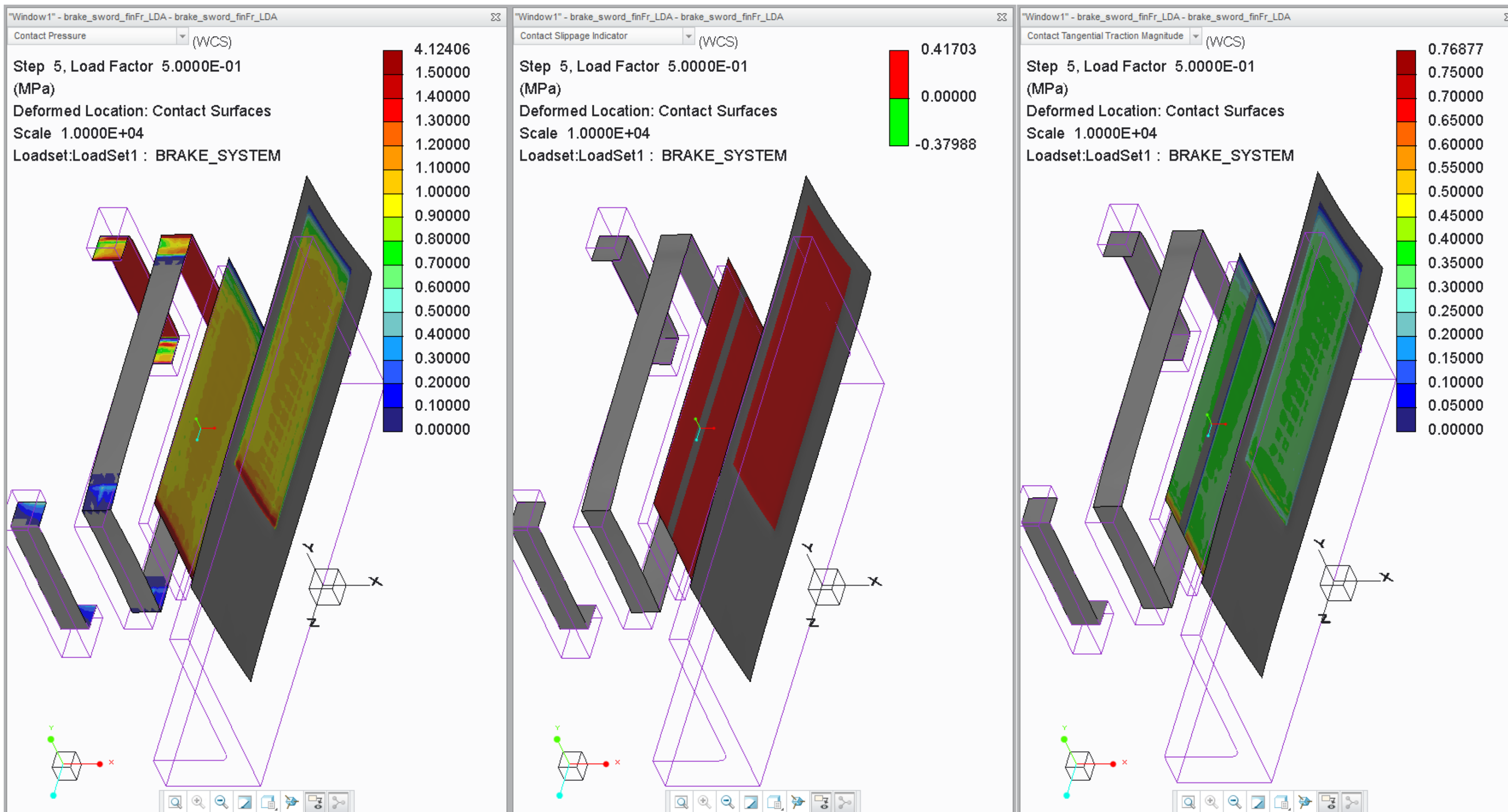
1. Brake system with brake pad and brake sword

1.4 Results | finite friction model | $\mu_{static}=\mu_{dynamic}=0.35$

***fully sliding

Load step 5: Clamping force 9,6 kN and 5 µm enforced sword displacement (scale 10000)

- System further fully slides (see messages in the engine *.pas file)



Exploded view!

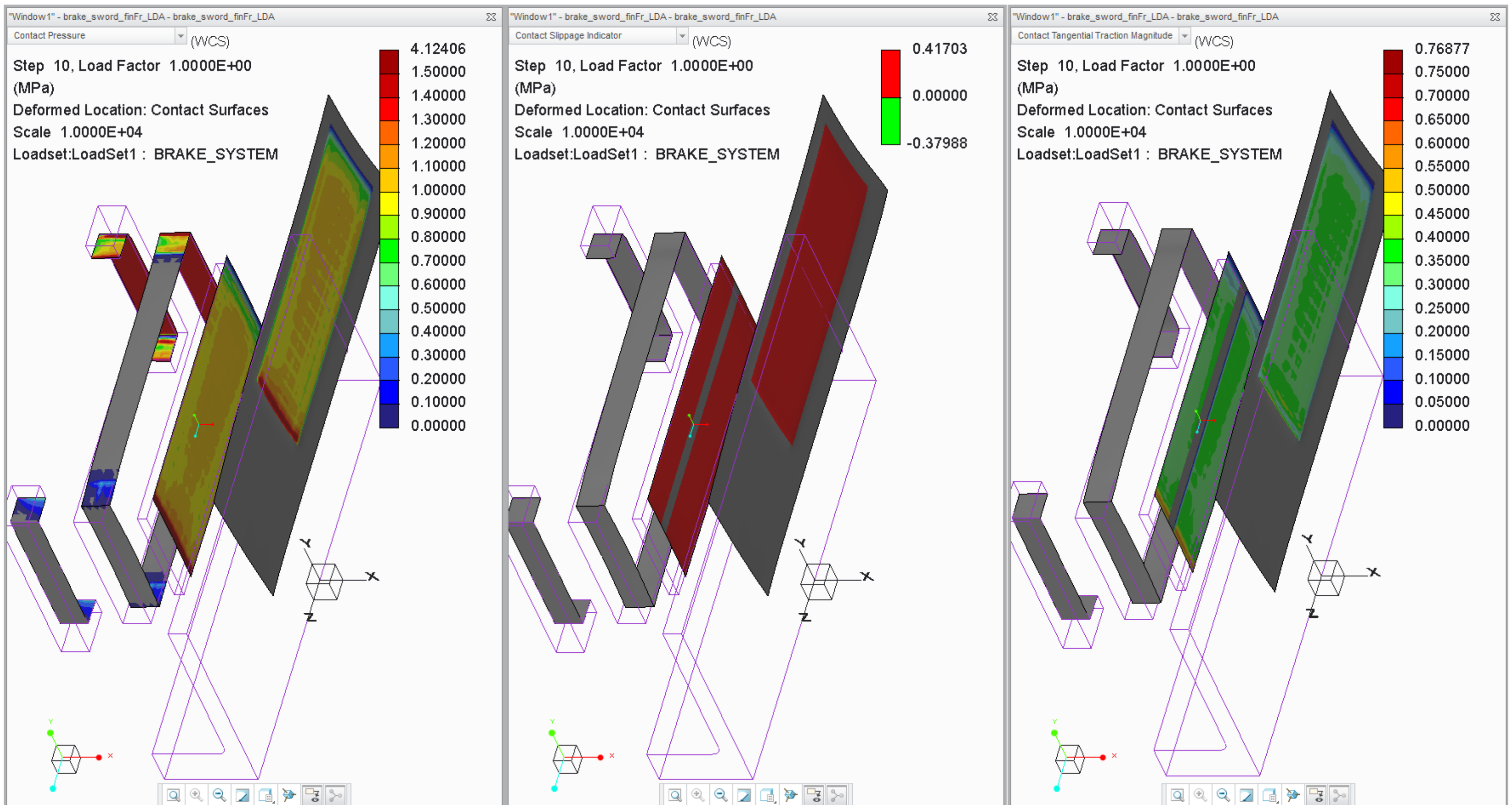
1. Brake system with brake pad and brake sword

1.4 Results | finite friction model | $\mu_{static}=\mu_{dynamic}=0.35$

***fully sliding

Load step 10: Clamping force 9,6 kN and 10 µm enforced sword displacement

- As expected, no further changes in loading, just fully sliding



Exploded view!

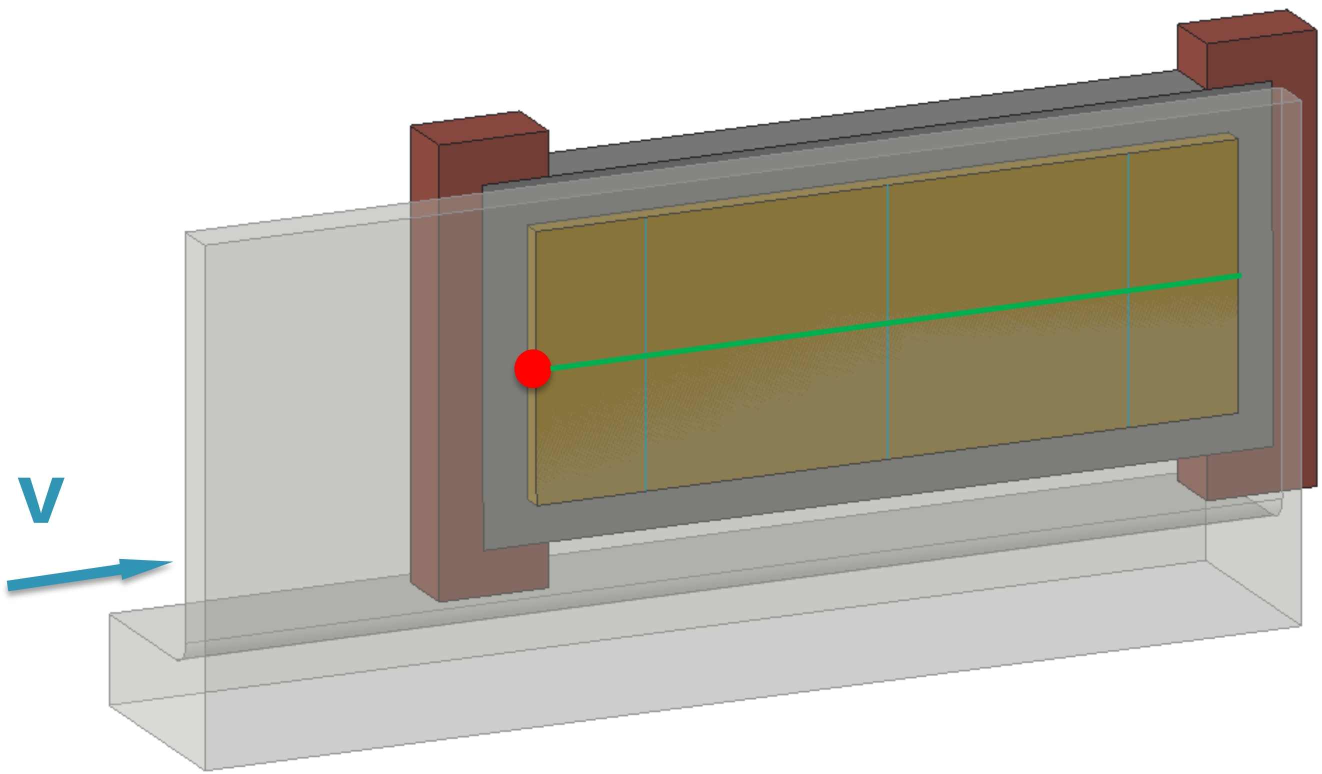
altran

1. Brake system with brake pad and brake sword

1.4 Results | finite friction model | $\mu_{static}=\mu_{dynamic}=0.35$

Result graph evaluation

- We now examine the physical quantities on the previously shown slides along the center line of the friction pad (shown as green line below)
- The origin of the following graphs therefore corresponds to the red marked point location of the evaluated line

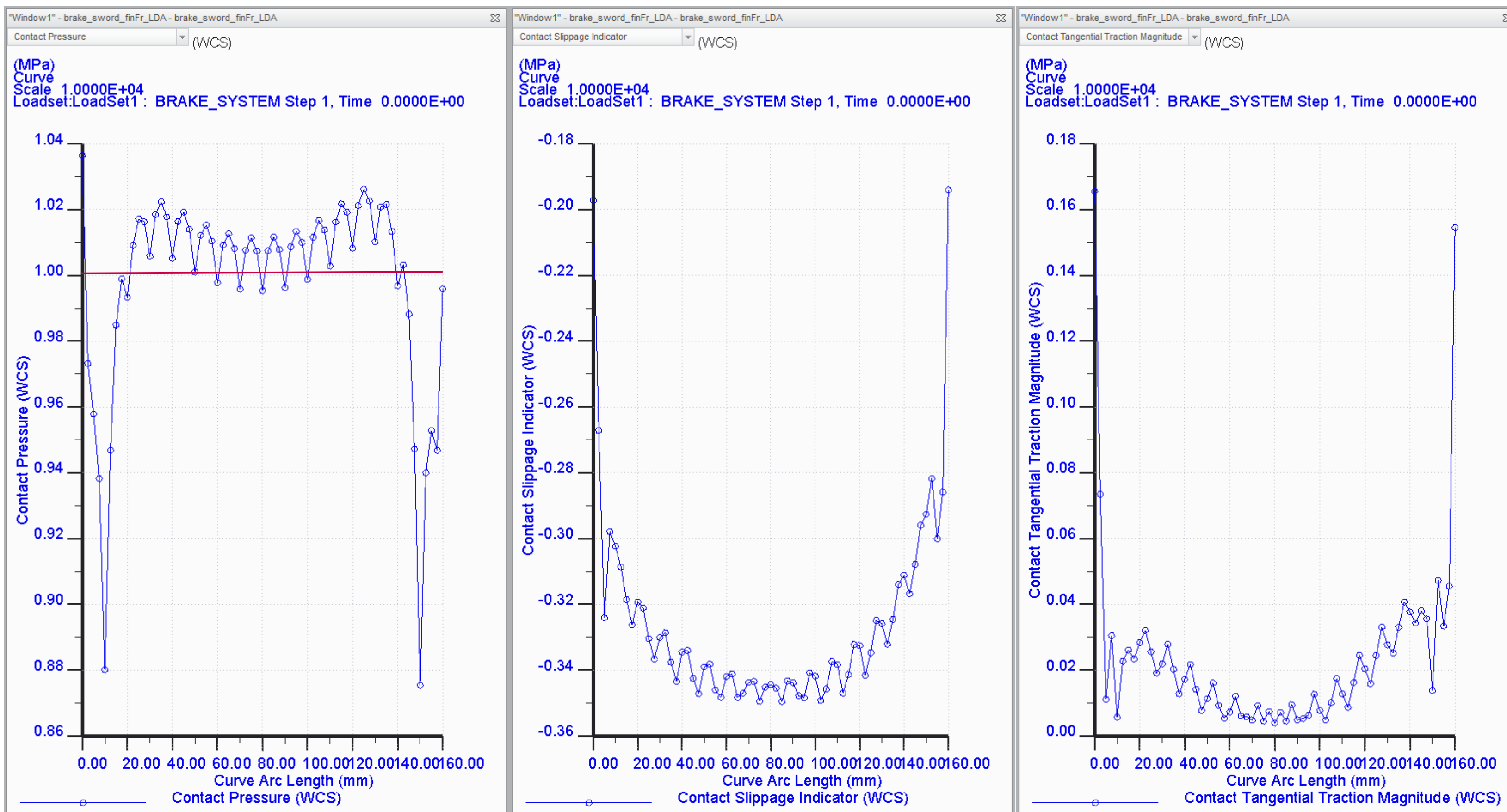


1. Brake system with brake pad and brake sword

1.4 Results | finite friction model | $\mu_{static}=\mu_{dynamic}=0.35$

Load step 0: Only clamping force 9,6 kN = Pressure 1 MPa applied

- Contact pressure [MPa]
- Contact slippage indicator [MPa]
- Contact tangential traction [MPa]



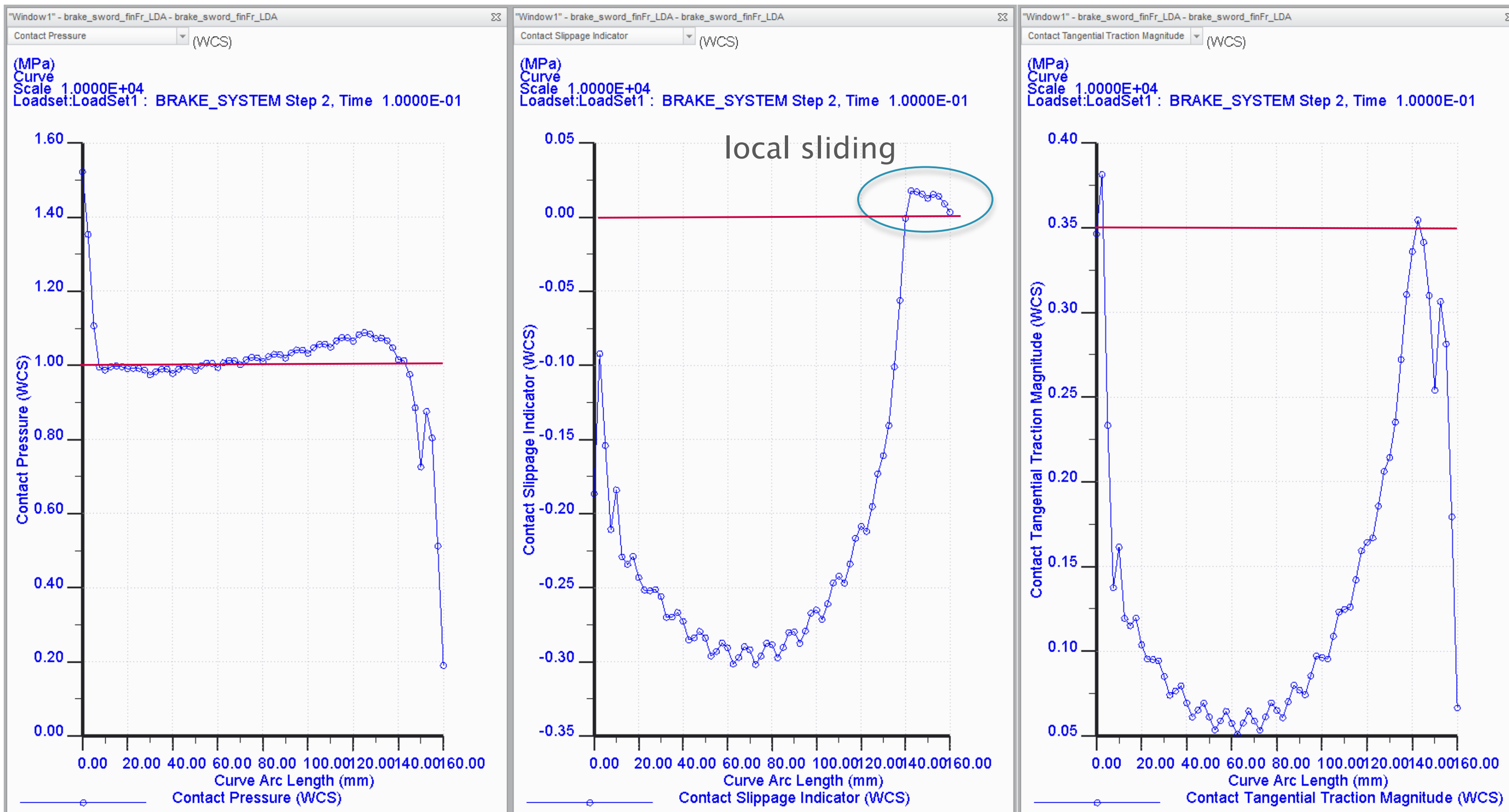
Very unsmooth results despite brick mesh –
SDA friction free contact does this much better!

1. Brake system with brake pad and brake sword

1.4 Results | finite friction model | $\mu_{static}=\mu_{dynamic}=0.35$

Load step 1: Clamping force 9,6 kN and 1 µm enforced sword displacement

- System deforms elastically, some local, but no global sliding



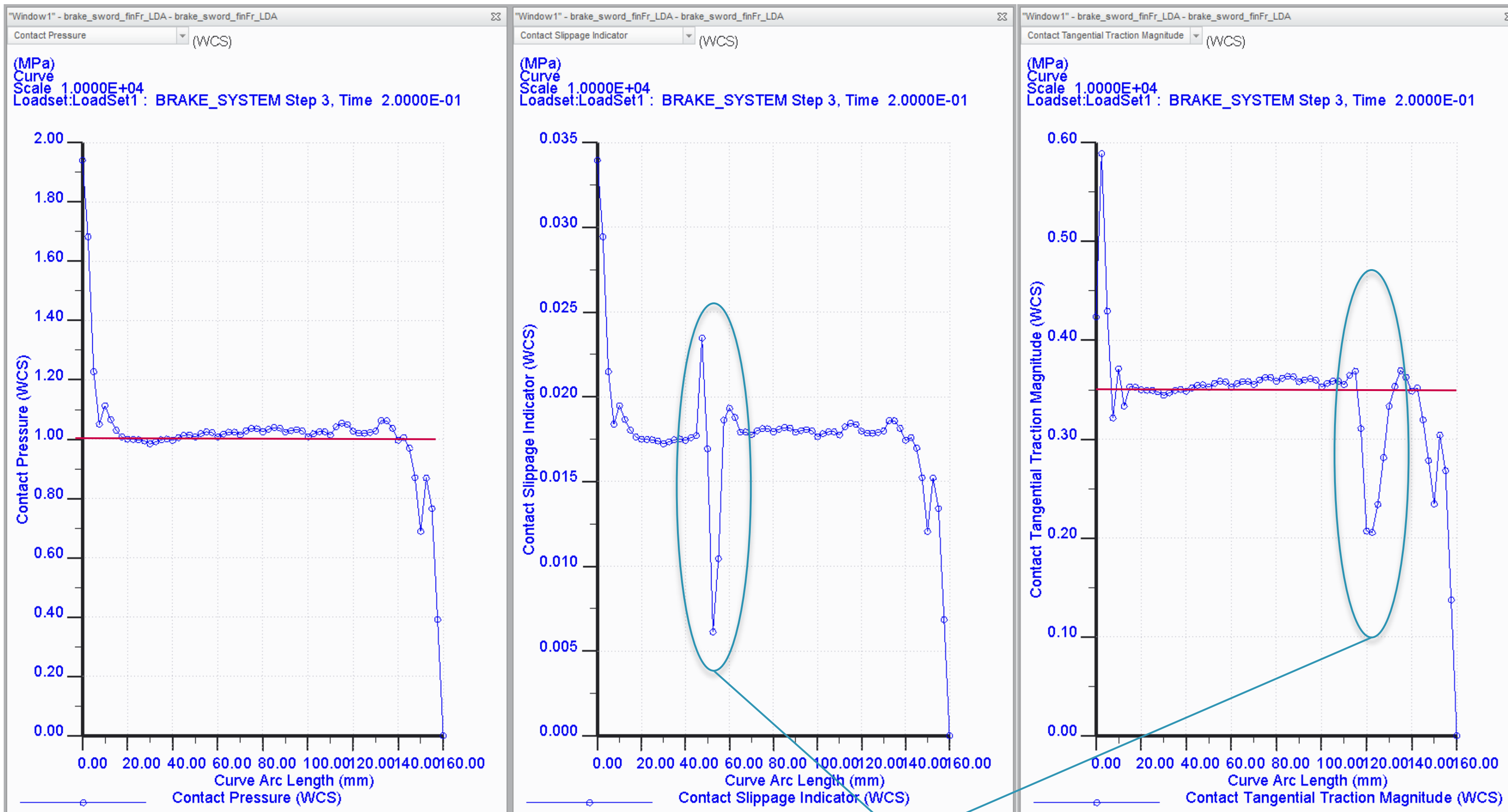
Minor Bug: In graph display (unlike in fringe display), Step number is shown +1

1. Brake system with brake pad and brake sword

1.4 Results | finite friction model | $\mu_{static}=\mu_{dynamic}=0.35$

Load step 2: Clamping force 9,6 kN and 2 µm enforced sword displacement

- System further deforms elastically, along this evaluation line the system fully slides



Inaccurate, unreasonable numerical results

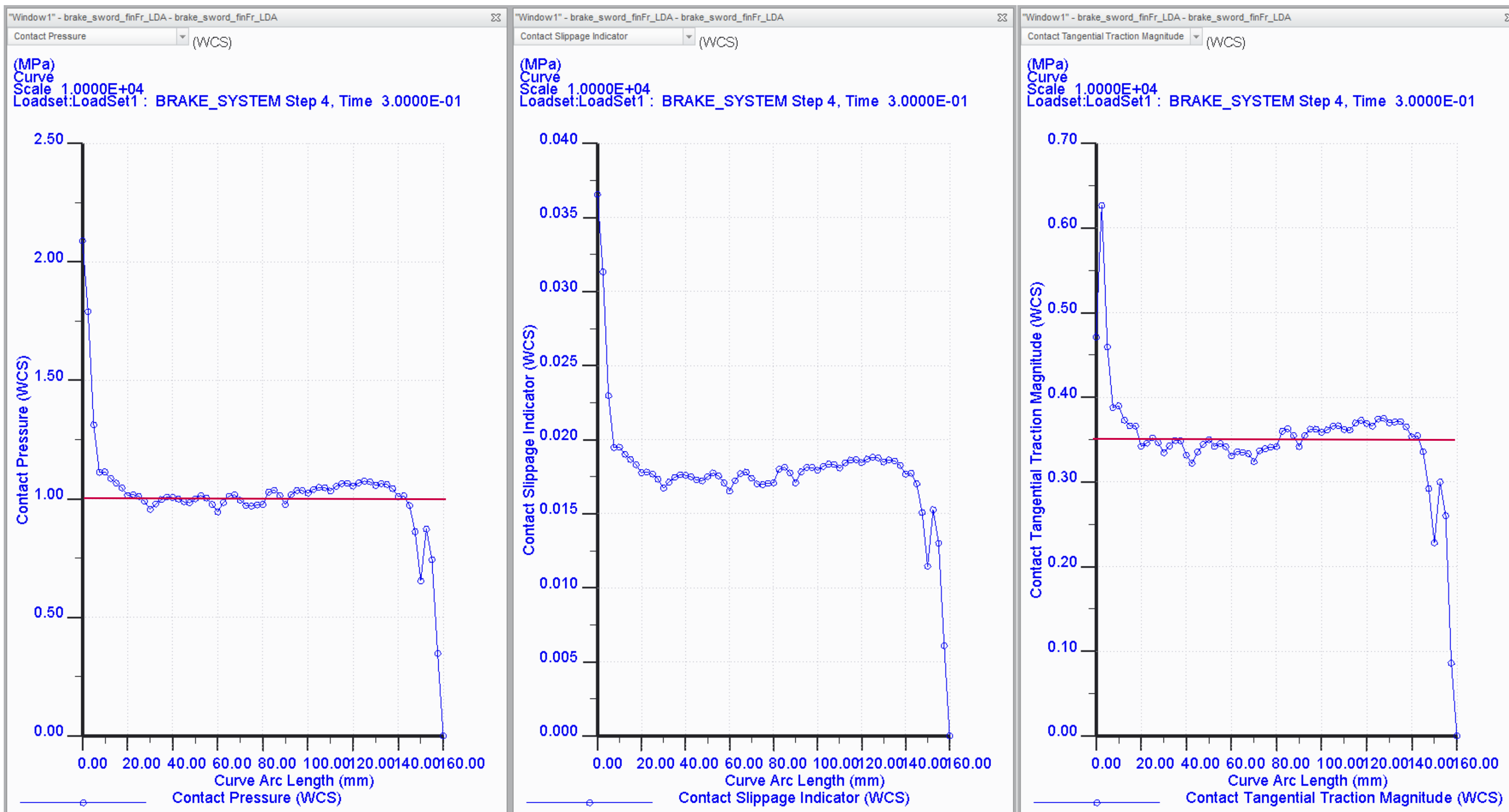
altran

1. Brake system with brake pad and brake sword

1.4 Results | finite friction model | $\mu_{static}=\mu_{dynamic}=0.35$

Load step 3: Clamping force 9,6 kN and 3 µm enforced sword displacement

- System fully slides

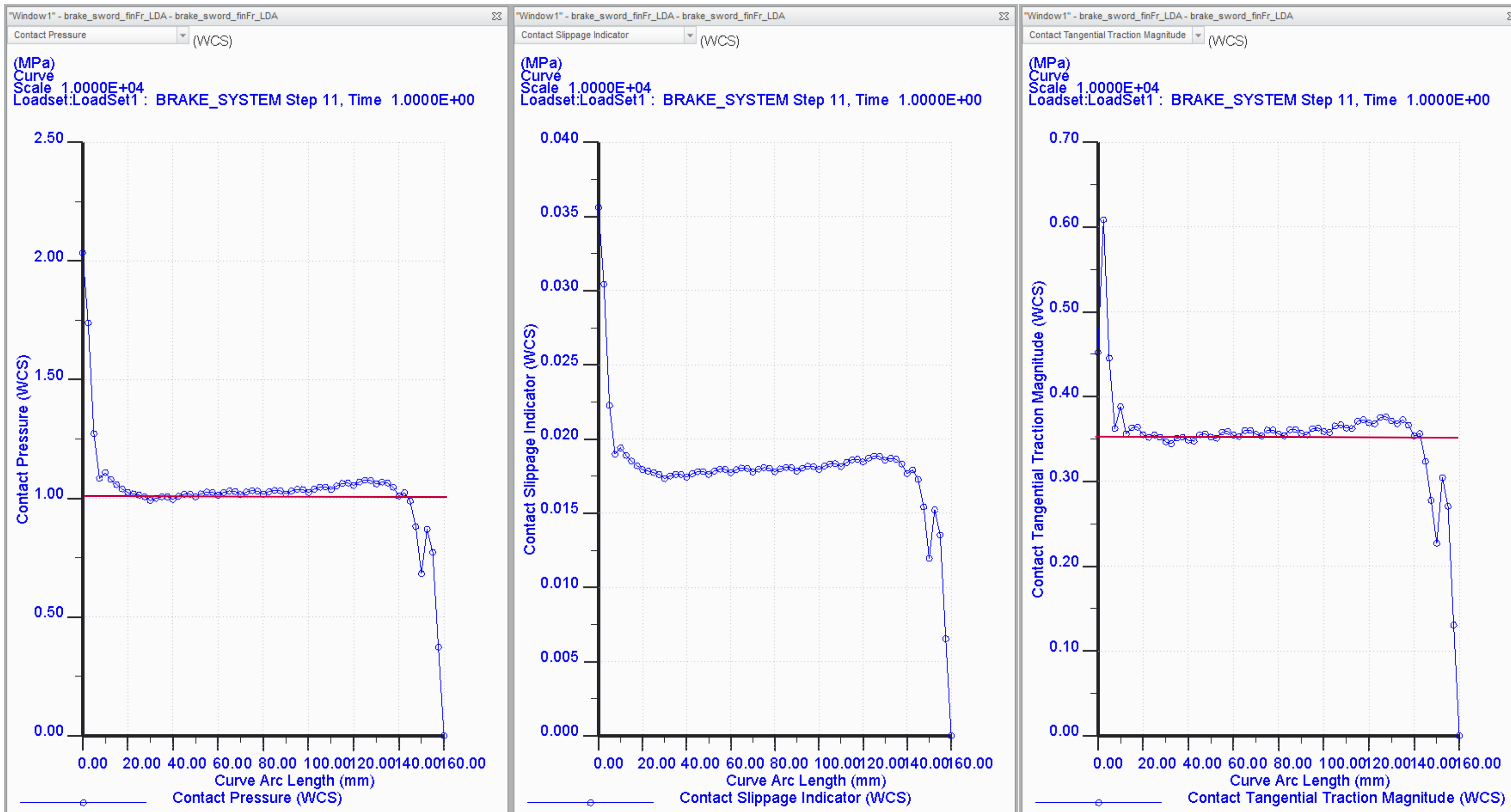


1. Brake system with brake pad and brake sword

1.4 Results | finite friction model | $\mu_{static}=\mu_{dynamic}=0.35$

Load step 10: Clamping force 9,6 kN and 10 µm enforced sword displacement

- As expected, no further changes in loading, just fully sliding

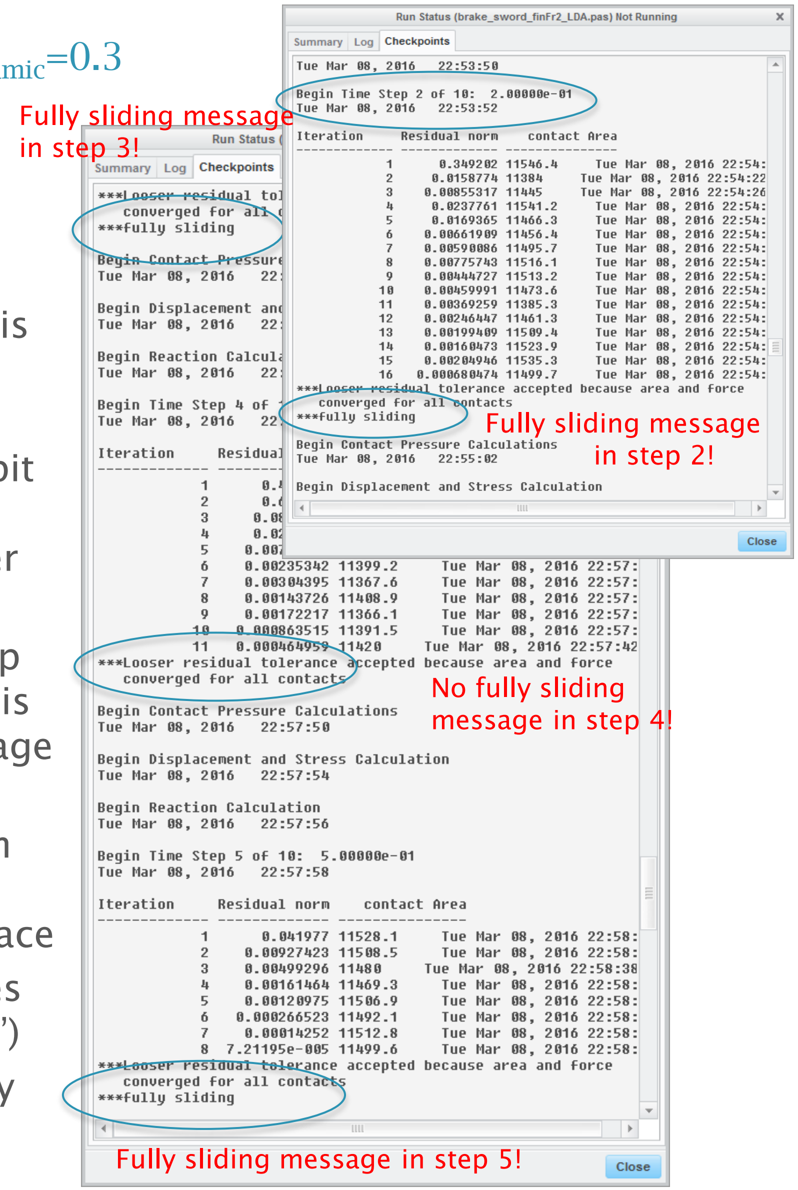


1. Brake system with brake pad and brake sword

1.5 Results | finite friction model | μ_{static}=0.35, $\mu_{dynamic}$=0.3

The lower dynamic friction coefficient should change results:

- After sliding appears, the tangential traction should drop from 0.35 to 0.3 MPa, since in this case the lower dynamic friction coefficient is active
- Therefore, the system should start to slide a bit earlier, since the contact pressure and local tangential traction is unevenly distributed over the contact surface
- The system reports fully sliding already in step 2 of 10, but (surprisingly) sticks in step 4 – this is not in line with the fringe plot for the slippage indicator of step 4 which shows fully sliding!
- In the result animation, we can see the system slips back in step 4, so here (or before) the change from static to dynamic sliding took place
- From step 5 on, the system permanently slides acc. to the *.pas-file messages ("Checkpoints")
- Note right: Accepted residual norms are pretty high and indicate inaccurate results (see SPR 4633631)



Rev. 1.1 | 24.03.2016

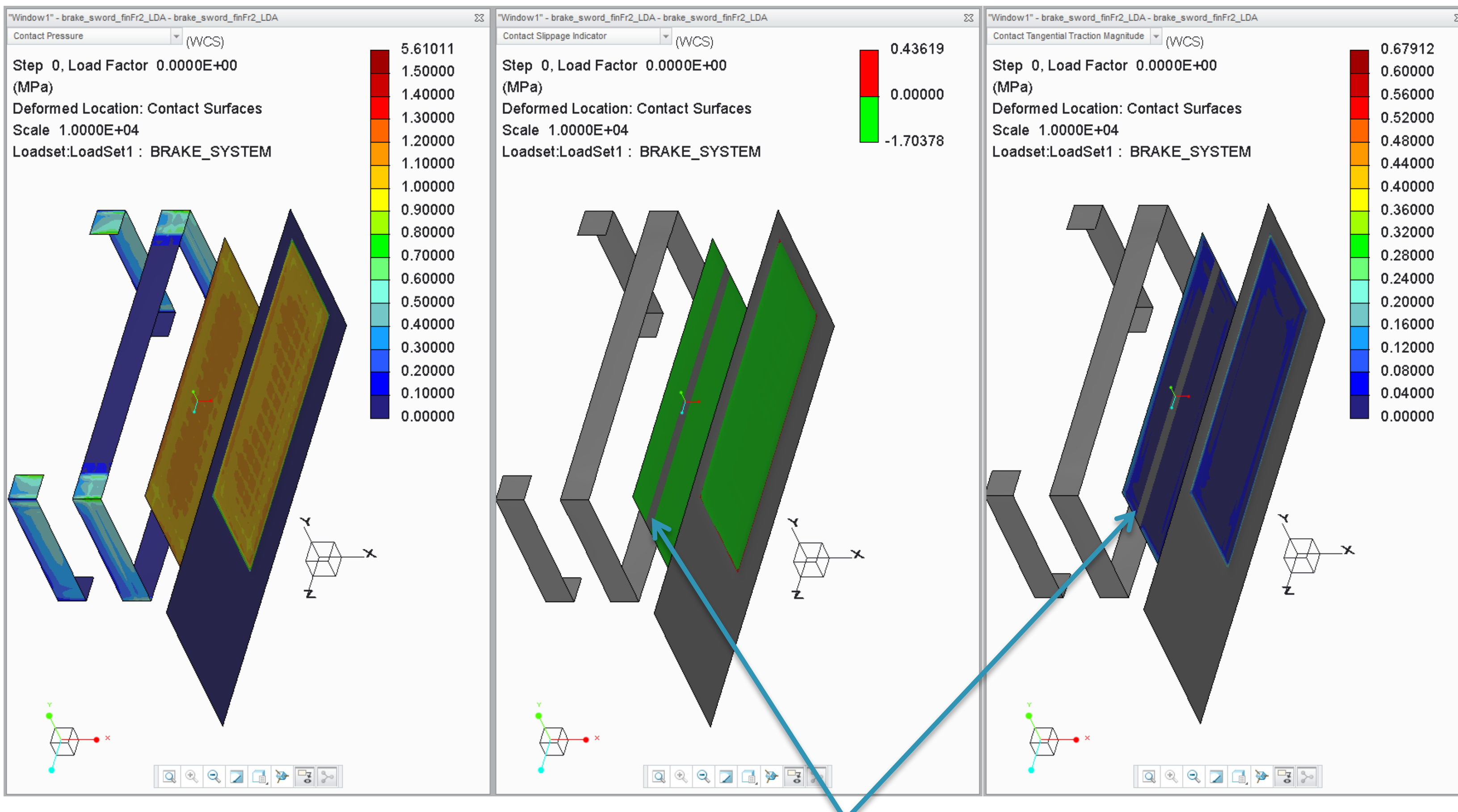
altran

1. Brake system with brake pad and brake sword

1.5 Results | finite friction model | μ_{static}=0.35, $\mu_{dynamic}$=0.3

Load step 0: Only clamping force 9,6 kN = Pressure 1 MPa applied (disp scale 10000)

- Contact pressure [MPa]
- Contact slippage indicator [MPa]
- Contact tangential traction [MPa]



Exploded view!

Again graphics bug – stripe with no results at contact surface

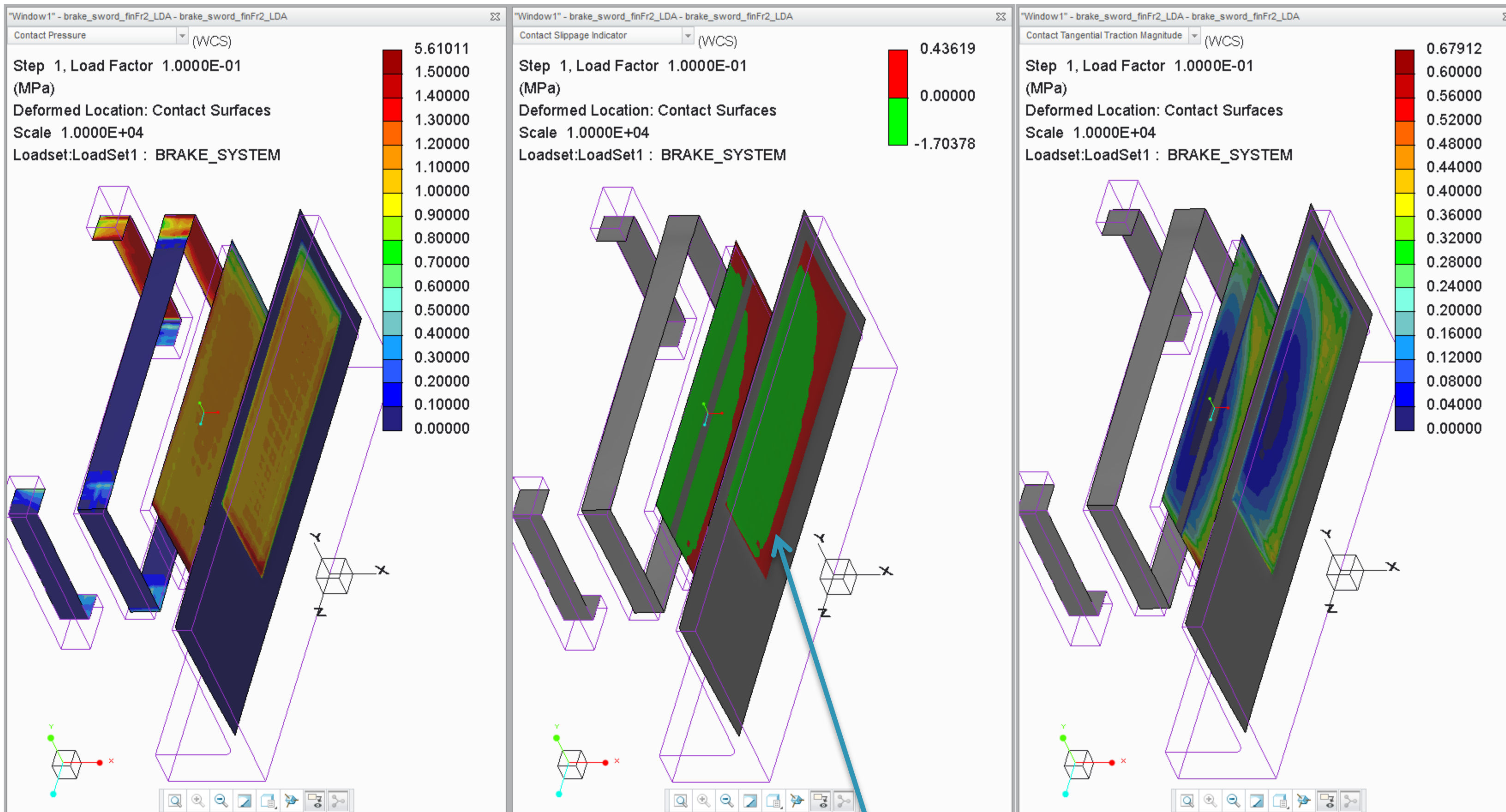
1. Brake system with brake pad and brake sword

1.5 Results | finite friction model | μ_{static}=0.35, $\mu_{dynamic}$=0.3

No fully sliding message in the pas-file!

Load step 1: Clamping force 9,6 kN and 1 µm enforced sword displacement (scale 10000)

- System deforms elastically, some local, but no global sliding



Exploded view!

Local sliding at red regions only

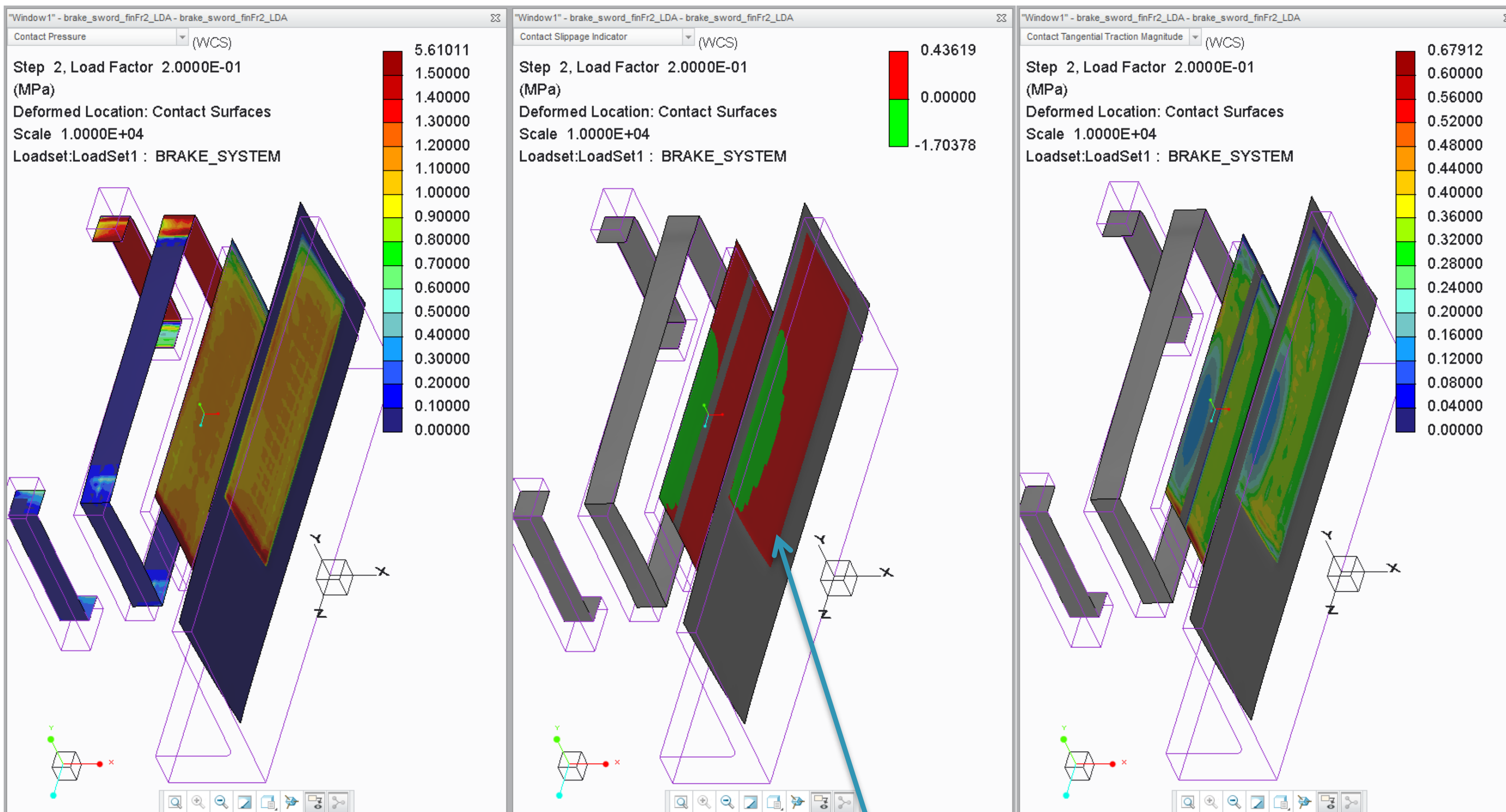
1. Brake system with brake pad and brake sword

1.5 Results | finite friction model | μ_{static}=0.35, $\mu_{dynamic}$=0.3

***fully sliding

Load step 2: Clamping force 9,6 kN and 2 µm enforced sword displacement (scale 10000)

- System further deforms elastically, some further local, but still no visible global sliding



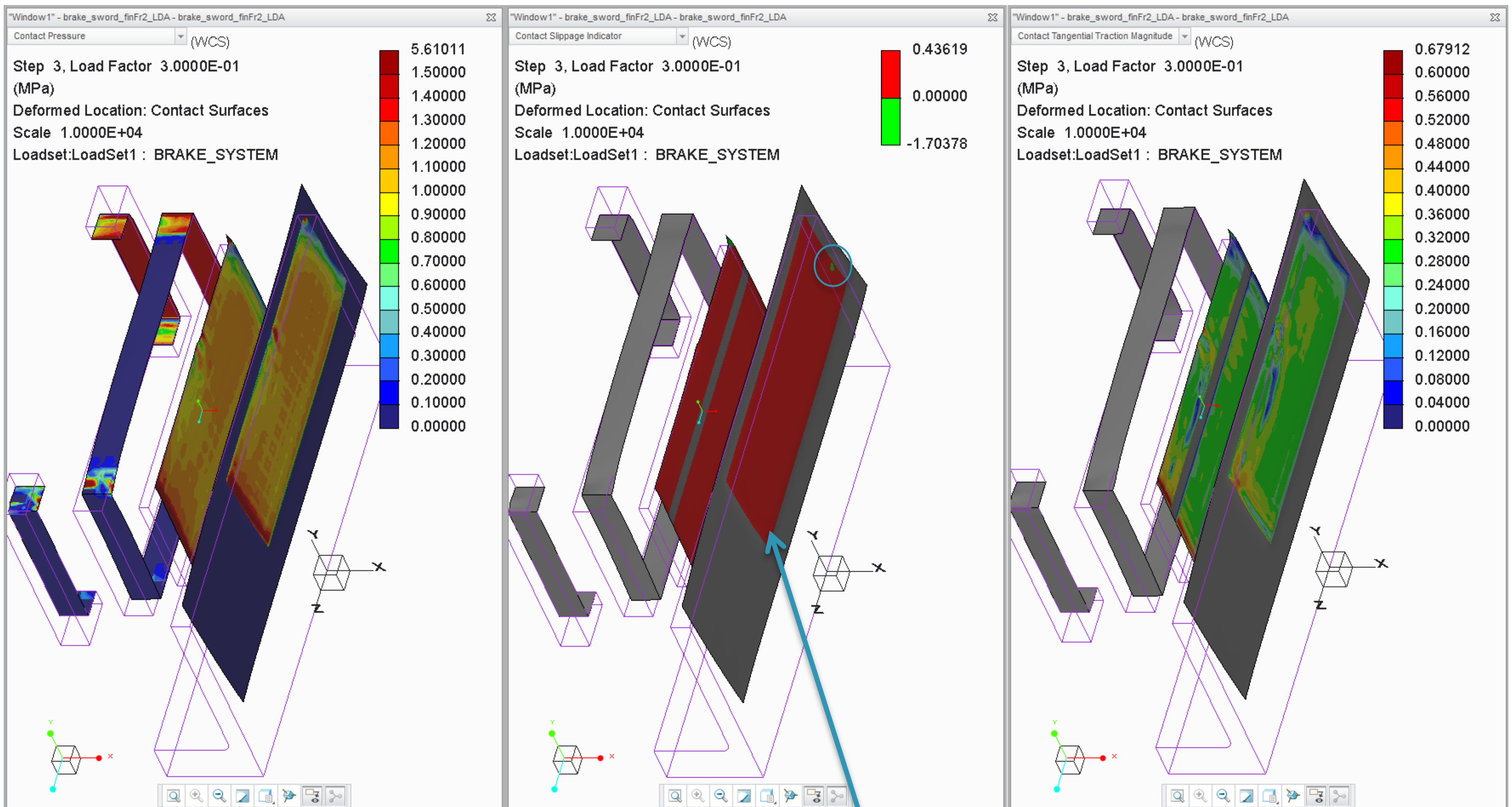
Exploded view! Local sliding at red regions only, but the pas-file reports fully sliding!

1. Brake system with brake pad and brake sword

1.5 Results | finite friction model | μ_{static}=0.35, $\mu_{dynamic}$=0.3

Load step 3: Clamping force 9,6 kN and 3 µm enforced sword displacement (scale 10000)

- System reports fully sliding, but still more elastic energy is stored



Exploded view!

Fully sliding, (nearly) all is red!

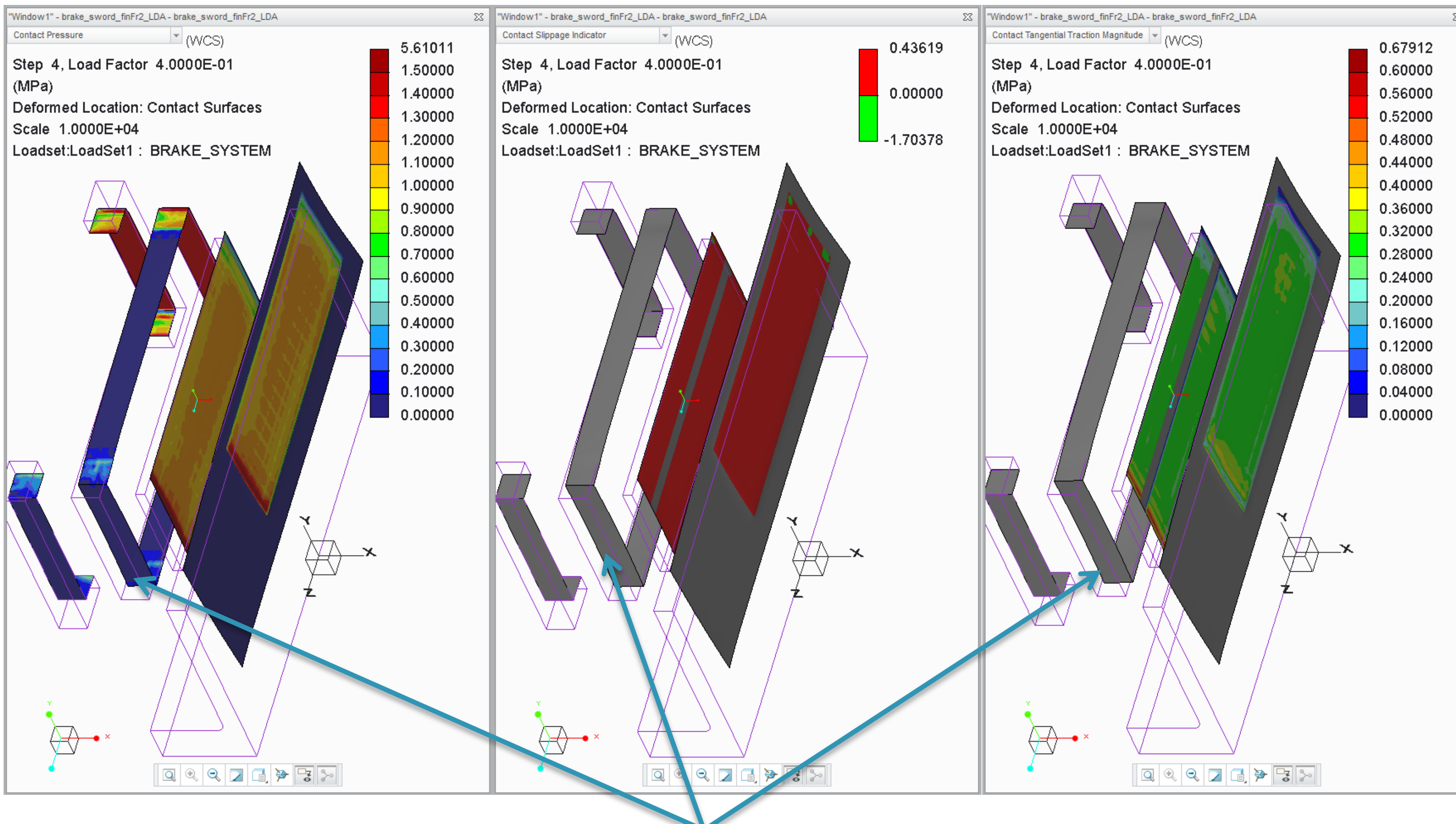
1. Brake system with brake pad and brake sword

1.5 Results | finite friction model | μ_{static}=0.35, $\mu_{dynamic}$=0.3

No fully sliding message in the pas-file!

Load step 4: Clamping force 9,6 kN and 4 µm enforced sword displacement (scale 10000)

- System slides back (elastic energy is released)



Exploded view!

Brake pad slides back after previously sticking

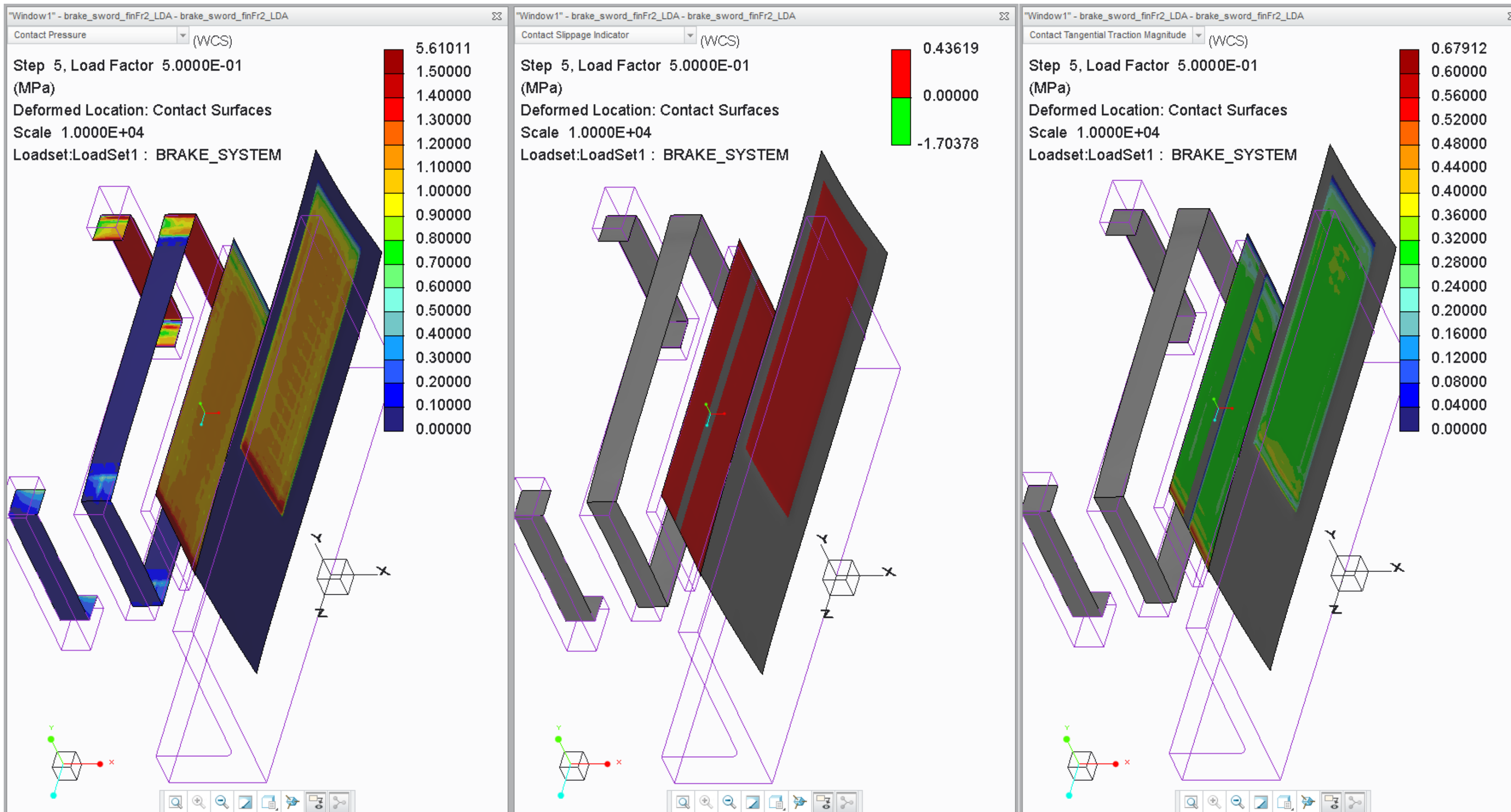
1. Brake system with brake pad and brake sword

1.5 Results | finite friction model | μ_{static}=0.35, $\mu_{dynamic}$=0.3

***fully sliding

Load step 5: Clamping force 9,6 kN and 5 µm enforced sword displacement (scale 10000)

- System further fully slides (no further elastic energy stored, brake pad stands still!)



Exploded view!

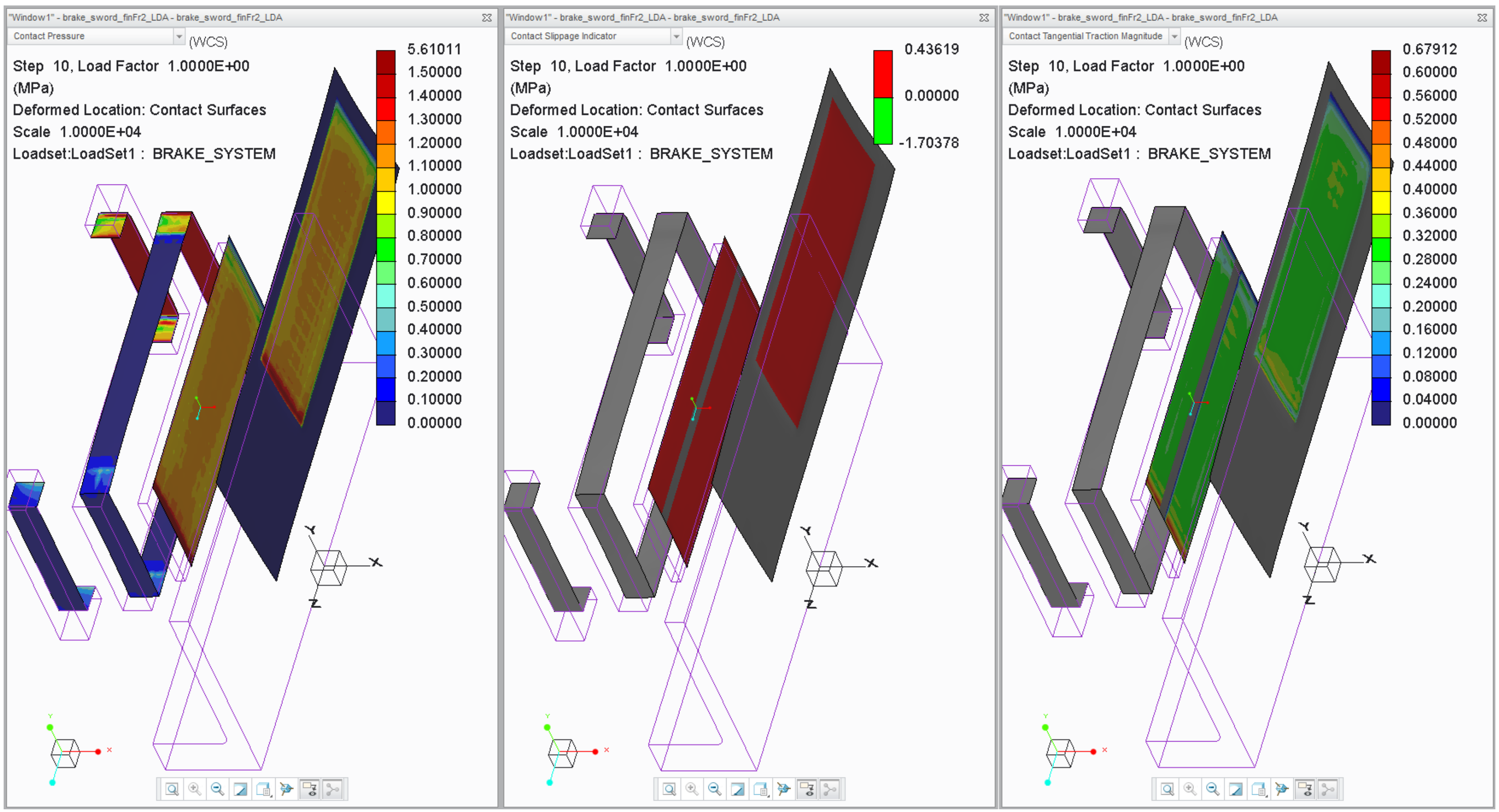
1. Brake system with brake pad and brake sword

1.5 Results | finite friction model | μ_{static}=0.35, $\mu_{dynamic}$=0.3

***fully sliding

Load step 10: Clamping force 9,6 kN and 10 µm enforced sword displacement

- As expected, no further changes in loading, just fully sliding



Exploded view!

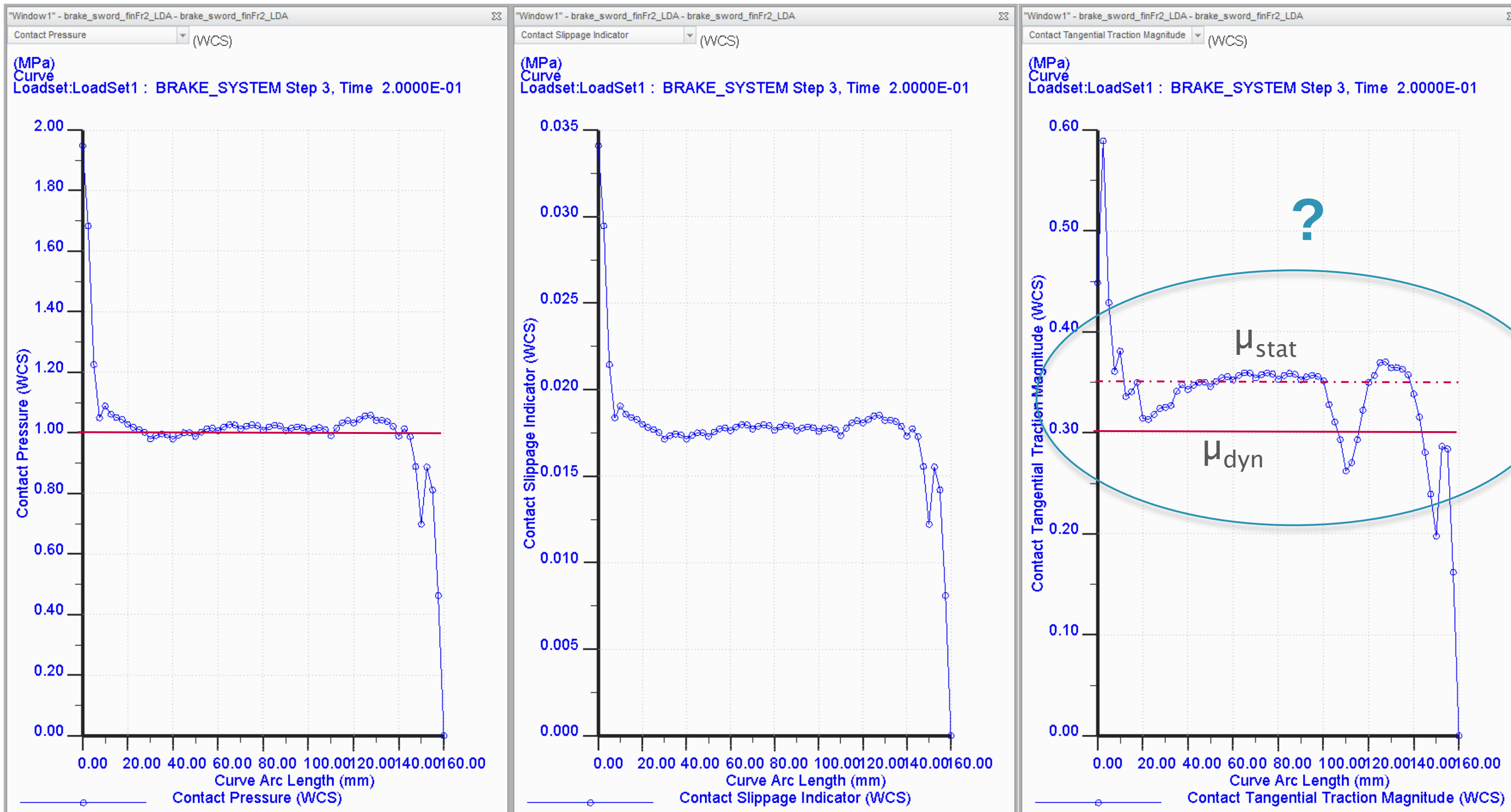
1. Brake system with brake pad and brake sword

1.5 Results | finite friction model | μ_{static}=0.35, $\mu_{dynamic}$=0.3

***fully sliding

Load step 2: Clamping force 9,6 kN and 2 µm enforced sword displacement

- .pas-file repots shows fully sliding, but fringe (and graph) results do not



altran

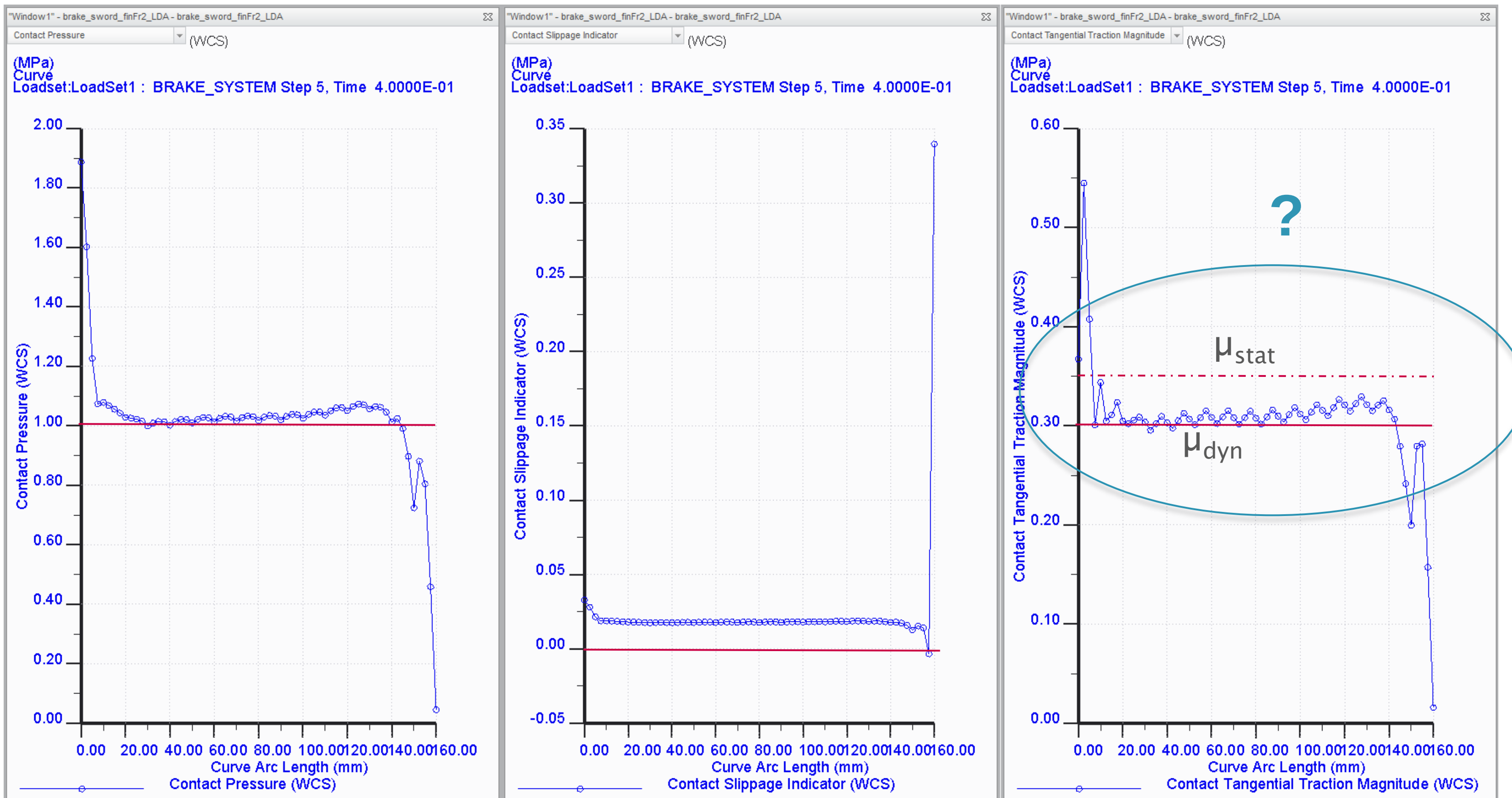
1. Brake system with brake pad and brake sword

1.5 Results | finite friction model | $\mu_{static}=0.35$, $\mu_{dynamic}=0.3$

No fully sliding message in the pas-file!

Load step 4: Clamping force 9,6 kN and 4 µm enforced sword displacement

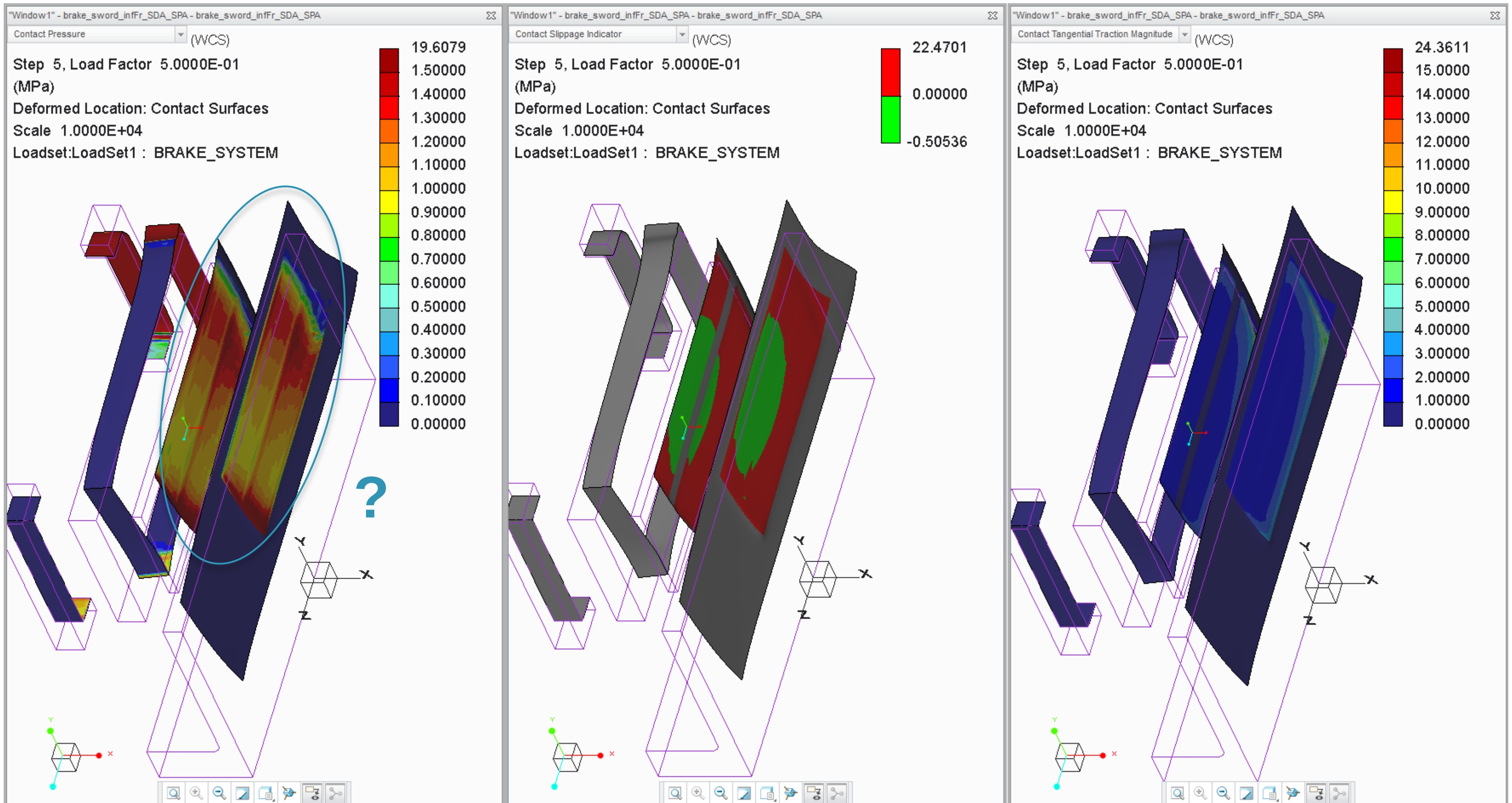
- Fringe results show fully sliding (slips back), but *pas-file does not report fully sliding



1. Brake system with brake pad and brake sword

1.6 Results | infinite friction model | μ_{static}=0.35

The infinite friction model shown below is not further evaluated here, since it shows wrong results – the contact pressure becomes totally unreasonable from load step 5 on (=enforced disp 5 µm):

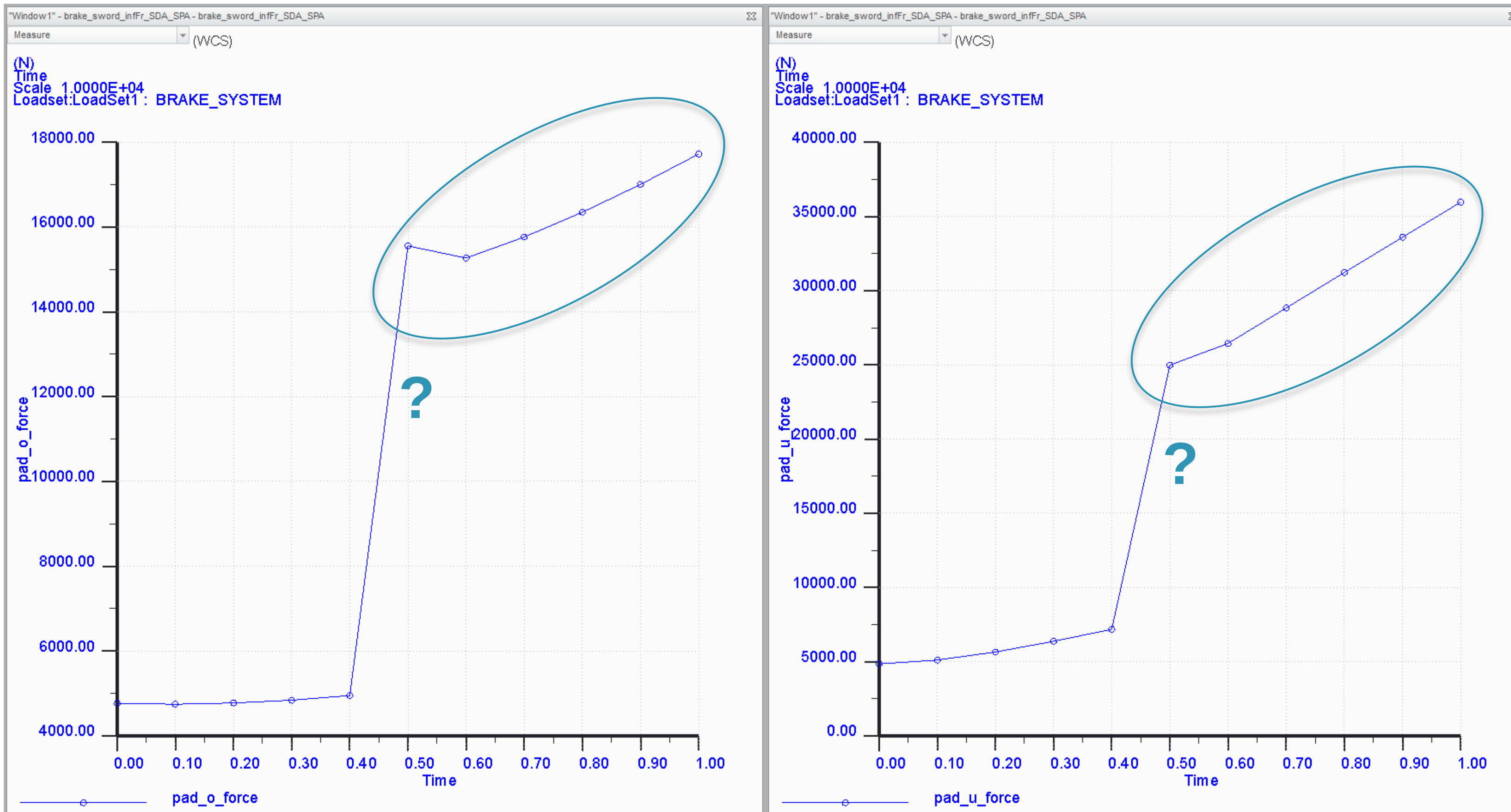


Exploded view!

1. Brake system with brake pad and brake sword

1.6 Results | infinite friction model | μ_{static}=0.35

In addition, the contact force transferred over the two half surfaces of the friction pad becomes totally wrong from load step 5 on (should be 9600 N in total at step 0) :

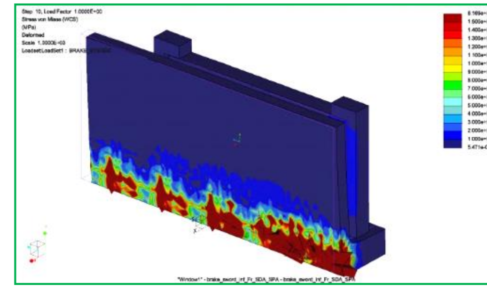


1. Brake system with brake pad and brake sword

1.7 Found issues during working on example 1

1.7.1 Infinite friction model

1. Mirror symmetry constraint at the sword does not work, it has to be replaced by standard constraint (see PTC Case 12966237) – the von Mises stress and local displacement results become totally wrong:

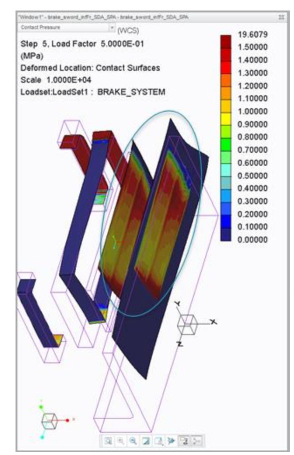


2. Extremely poor convergence (up to 368 iterations per load step, >12 h elapsed time and >58 h CPU time (eight core PC used!!)

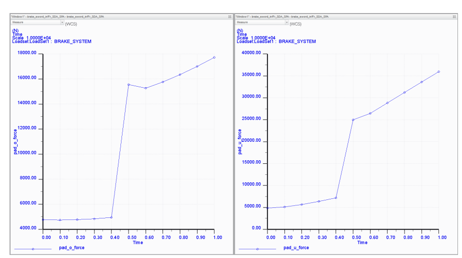
```
364        7.2066e-1.
365        4.1149e-12     1.0455e+04   Tue Mar 08, 2016    21:49:11
366        2.3343e-12     1.0455e+04   Tue Mar 08, 2016    21:49:13
367        1.3157e-12     1.0455e+04   Tue Mar 08, 2016    21:49:14
368        7.3680e-13     1.0455e+04   Tue Mar 08, 2016    21:49:16

Begin Contact Postprocessing
  Mar 08, 2016    21:49:16
```

```
Total Elapsed Time (seconds): 45052.56
Total CPU Time     (seconds): 210280.05
Maximum Memory Usage (kilobytes): 9283217
Working Direct      Usage (kilobytes)
```

3. Wrong contact pressure results at the friction pad starting with load step 5



4. Wrong contact force measure results at the friction pad

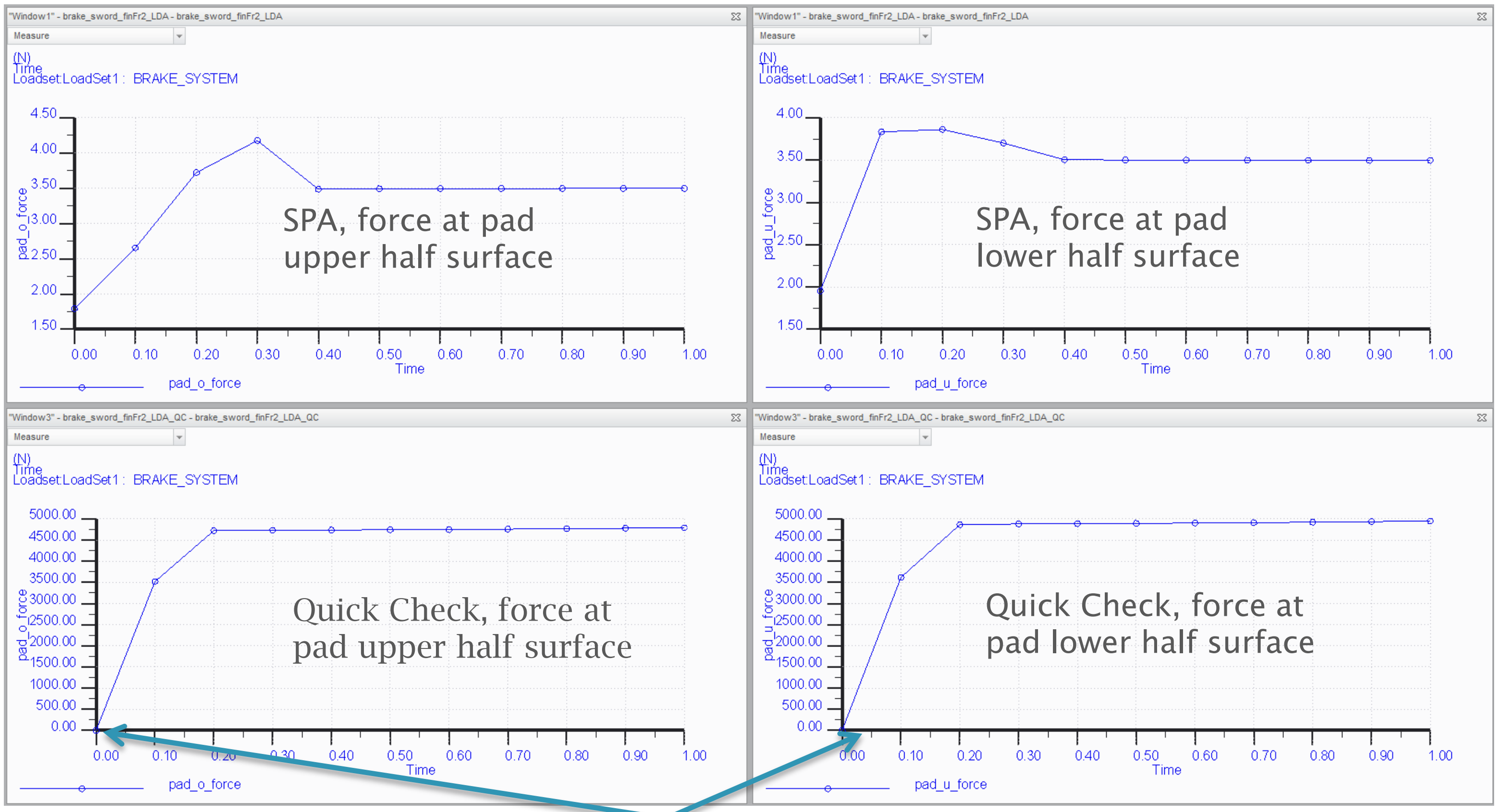


1. Brake system with brake pad and brake sword

1.7 Found issues during working on example 1

1.7.2 Finite friction model

1. Contact force measures deliver totally wrong results in SPA convergence, they just work partially in Quick Check (remember expected value for both together is around 9600 N):



Already at load step 0, the full force of 9600 N should be reported (and not Zero!), as fringe contact pressure results correctly indicate!

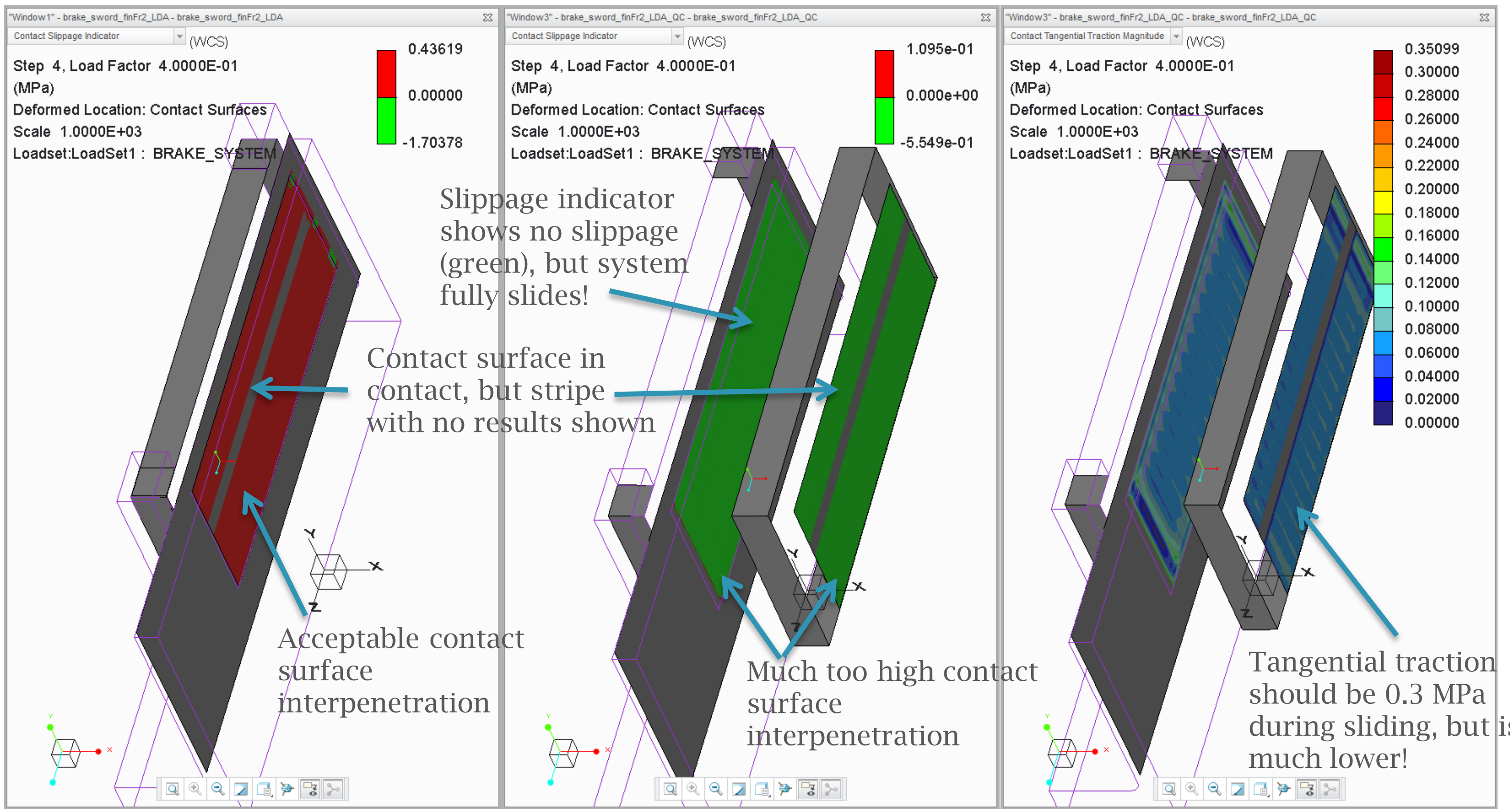
altran

1. Brake system with brake pad and brake sword

1.7 Found issues during working on example 1

1.7.2 Finite friction model

2. In Quick Check convergence, therefore many other fringe results become wrong (left SPA results, middle and right QC, disp. mag. factor 1000):



1. Brake system with brake pad and brake sword

1.7 Found issues during working on example 1

1.7.2 Finite friction model

3. Neither in Quick check, nor in SPA, there is ever an update in penalty spring stiffness during the iterations for this model (default settings used). This is at least unusual (see right).

```
Run Status (brake_sword_finFr2_LDA_QC.pas) Not Running
Summary  Log  Checkpoints

Iteration      Residual norm    contact Area
------------- ------------- ---------------
            1   0.000743004 11088.7     Tue Mar 08, 2016 23:41:21
            2   4.8087e-007 11088.7     Tue Mar 08, 2016 23:41:27
            3     1.33e-016 11088.7     Tue Mar 08, 2016 23:41:32

Begin Contact Pressure Calculations
Tue Mar 08, 2016   23:41:34

Begin Displacement and Stress Calculation
Tue Mar 08, 2016   23:41:34

Begin Reaction Calculation
Tue Mar 08, 2016   23:41:36

Begin Time Step 8 of 10:  8.00000e-01
Tue Mar 08, 2016   23:41:37

Iteration      Residual norm    contact Area
------------- ------------- ---------------
            1   0.000570012 11093.3     Tue Mar 08, 2016 23:41:40
            2  8.65976e-009 11093.3     Tue Mar 08, 2016 23:41:45
            3   5.8856e-017 11093.3     Tue Mar 08, 2016 23:41:51

Begin Contact Pressure Calculations
Tue Mar 08, 2016   23:41:53

Begin Displacement and Stress Calculation
Tue Mar 08, 2016   23:41:54

Begin Reaction Calculation
Tue Mar 08, 2016   23:41:58

Begin Time Step 9 of 10:  9.00000e-01
Tue Mar 08, 2016   23:41:58

Iteration      Residual norm    contact Area
------------- ------------- ---------------
            1     0.0203124 11087.5     Tue Mar 08, 2016 23:42:01
            2   0.000158979 11086.1     Tue Mar 08, 2016 23:42:04
            3  1.37923e-006 11085.4     Tue Mar 08, 2016 23:42:10
            4  5.43548e-009 11085.4     Tue Mar 08, 2016 23:42:15
            5  5.39828e-017 11085.4     Tue Mar 08, 2016 23:42:21

Begin Contact Pressure Calculations
Tue Mar 08, 2016   23:42:22

Begin Displacement and Stress Calculation
Tue Mar 08, 2016   23:42:23

Begin Reaction Calculation
Tue Mar 08, 2016   23:42:25

Begin Time Step 10 of 10:  1.00000e+00
Tue Mar 08, 2016   23:42:26

Iteration      Residual norm    contact Area
------------- ------------- ---------------
            1     0.0128886 11089.9     Tue Mar 08, 2016 23:42:29
            2   0.000154286 11089.2     Tue Mar 08, 2016 23:42:34
            3  3.69261e-007 11089.2     Tue Mar 08, 2016 23:42:40
            4  1.10267e-015 11089.2     Tue Mar 08, 2016 23:42:45

Close
```

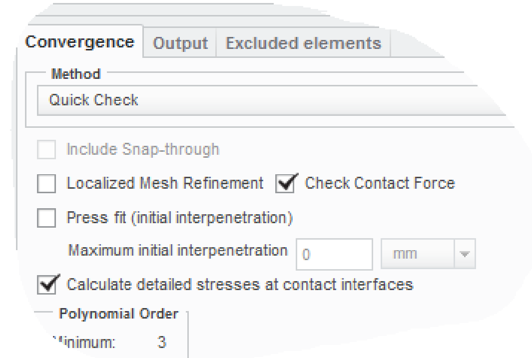
4. Using the engine command line option
–contactSpringRatio 100.0
drastically increases number of iterations, but does not improve this situation (still high interference):

```
69  4.77...
70  3.08529e-007 10512..
71  2.35907e-005 10546     Wed Mar 09, 20..
72   1.3945e-005 10612     Wed Mar 09, 2016
73  4.62371e-006 10913.4    Wed Mar 09, 2016
74  6.00147e-007 10953.7    Wed Mar 09, 2016 0..
75  4.17844e-007 10697.9    Wed Mar 09, 2016 00:2..
76  8.24897e-006 10713.1    Wed Mar 09, 2016 00:22:44
77    1.182e-006 10870.1    Wed Mar 09, 2016 00:22:54
78  6.19708e-007 10920.8    Wed Mar 09, 2016 00:23:00
79  1.66792e-006 10972.2    Wed Mar 09, 2016 00:23:05
80  3.56056e-007 10780.8    Wed Mar 09, 2016 00:23:13

Complete sliding was detected at one or more contacts
in your model. This can cause excessive motion and/or
poor convergence. Please review your loads and constraints.
```

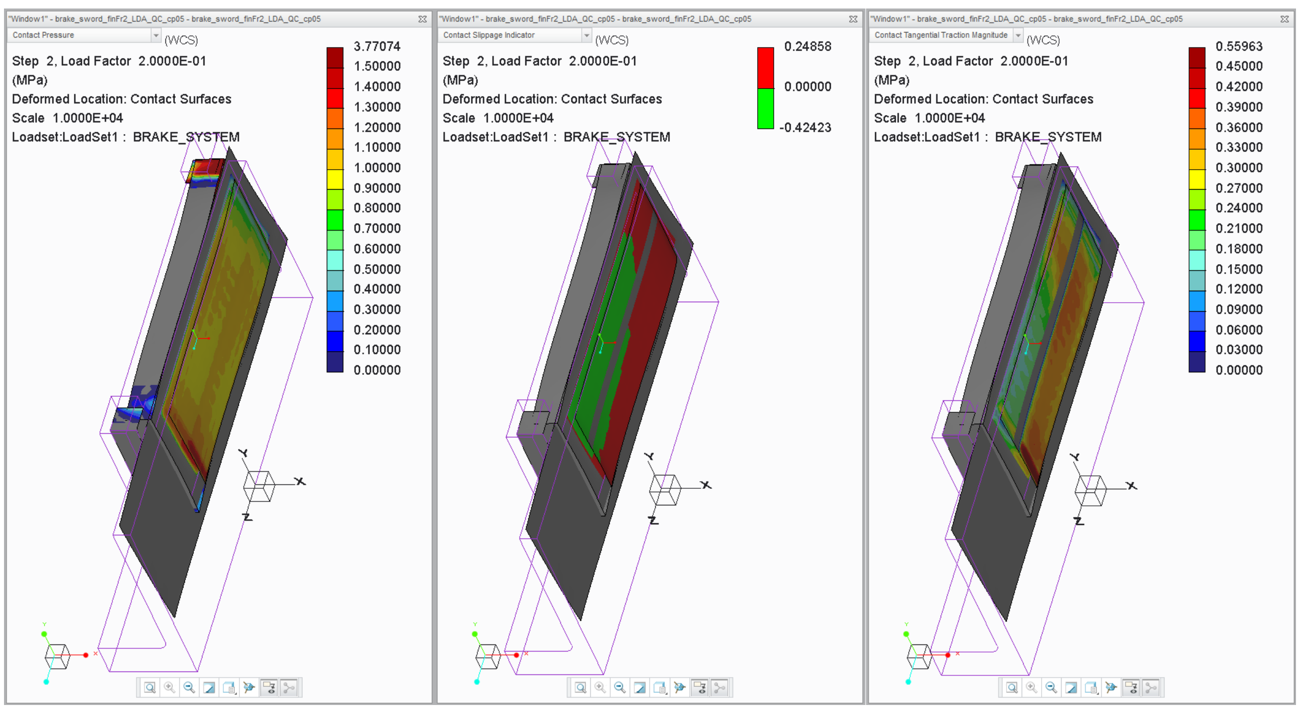
1. Brake system with brake pad and brake sword

1.7 Found issues during working on example 1

1.7.2 Finite friction model



5. Neither using “check contact force” nor “calculate detailed stresses at contact interfaces” improves quick check result quality of this model
6. Just using the engine command line option
   -contact_penetration 0.05
   in the Quick Check analysis helps to reduce interpenetration at the contacts and to obtain better results for slippage indicator and contact tangential traction magnitude:

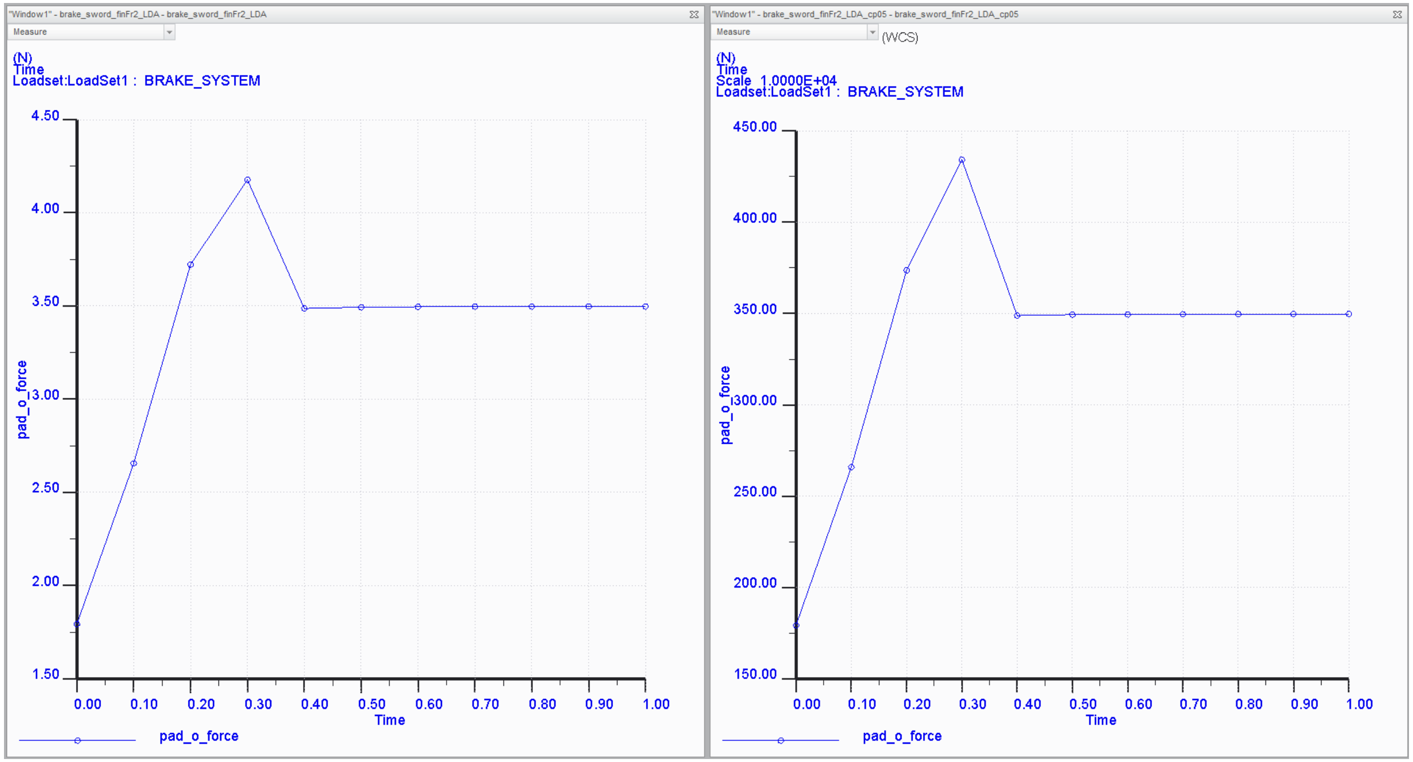


1. Brake system with brake pad and brake sword

1.7 Found issues during working on example 1

1.7.2 Finite friction model

7. Using the engine command line option
   -contact_penetration 0.05 (default 5, so factor 100 decreased)
   in an SPA instead of a Quick Check analysis, surprisingly increases the (totally wrong) contact force measure exactly with factor 100 (which is still wrong, since we expect values around 5000 N here):

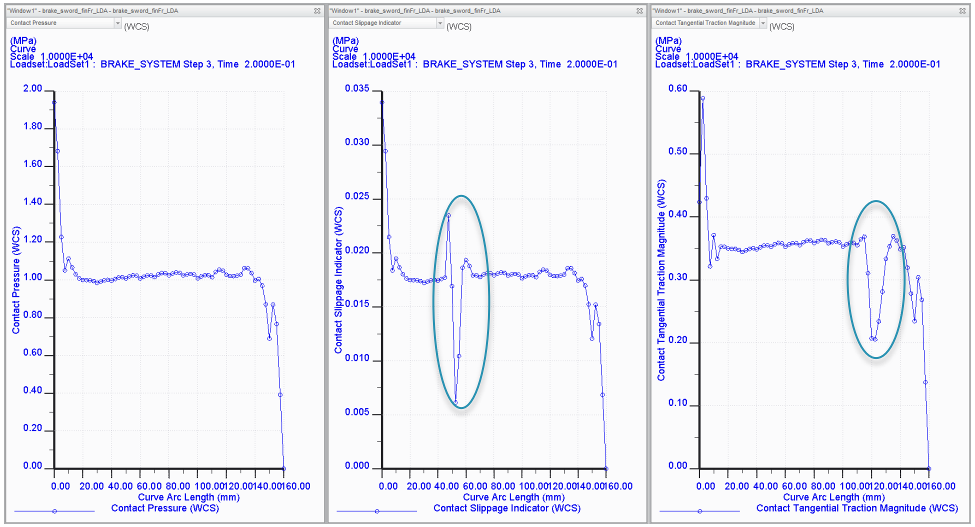


1. Brake system with brake pad and brake sword

1.7 Found issues during working on example 1

1.7.2 Finite friction model

8. Numerical quality of the results is often pretty poor



9. For the model with lower dynamic than static friction, results and messages for sliding/sticking appear to be questionable/inconsistent
10. In an SPA analysis, fully sliding messages are not reported in the pas-file in pass 1, only in pass 2 (and do not always seem to be reasonable)

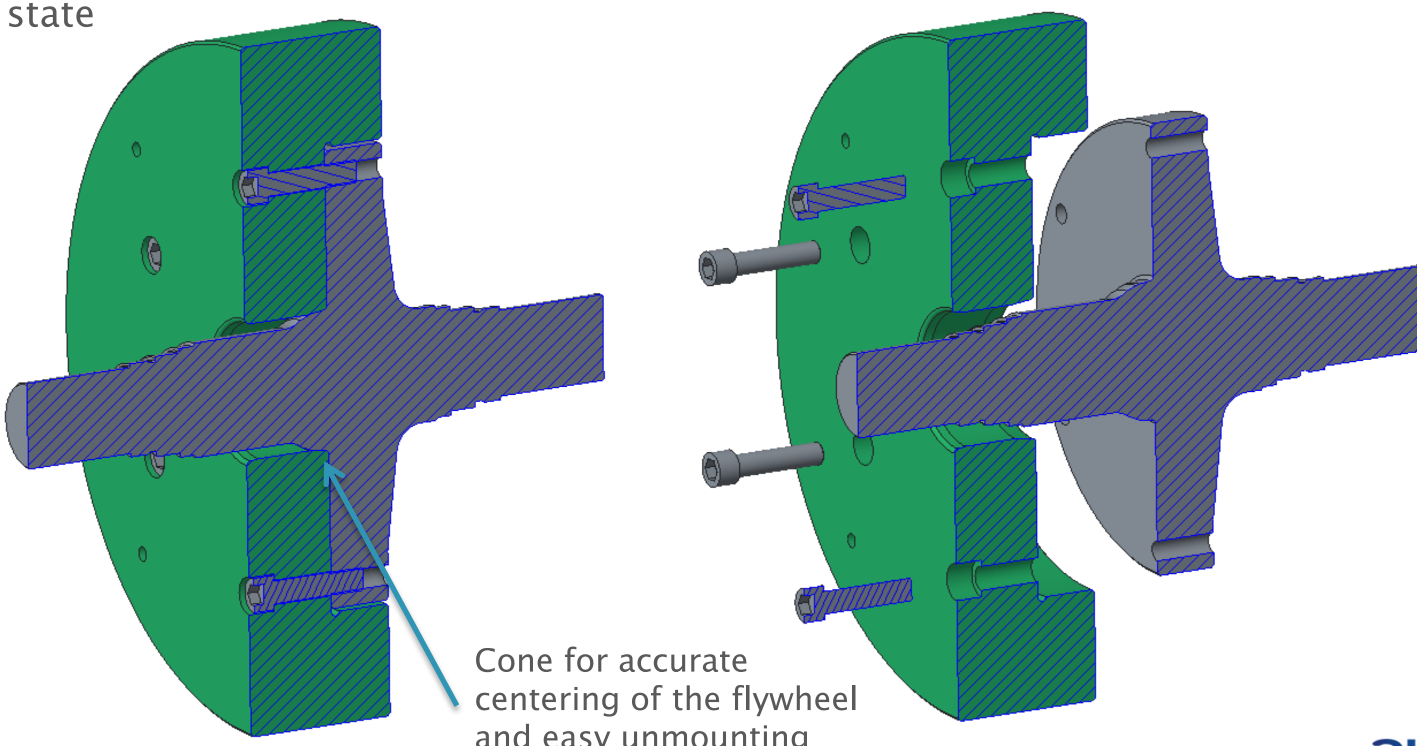
2. Flywheel with a bolted conical hub-shaft-connection

2.1 CAD model & problem description

Fly wheel using axial bolts for clamping and cone for accurate centering

Technical Data:

- 6 steel bolts M16x70, preloaded with 100 kN each
- Max. rotational speed 6000 rpm, Flywheel diameter approx. 500 mm
- Goal is to check if sliding appears under rotational loads at the flange from different relative strains at shaft and flywheel, and to study the resulting influence to the stress state

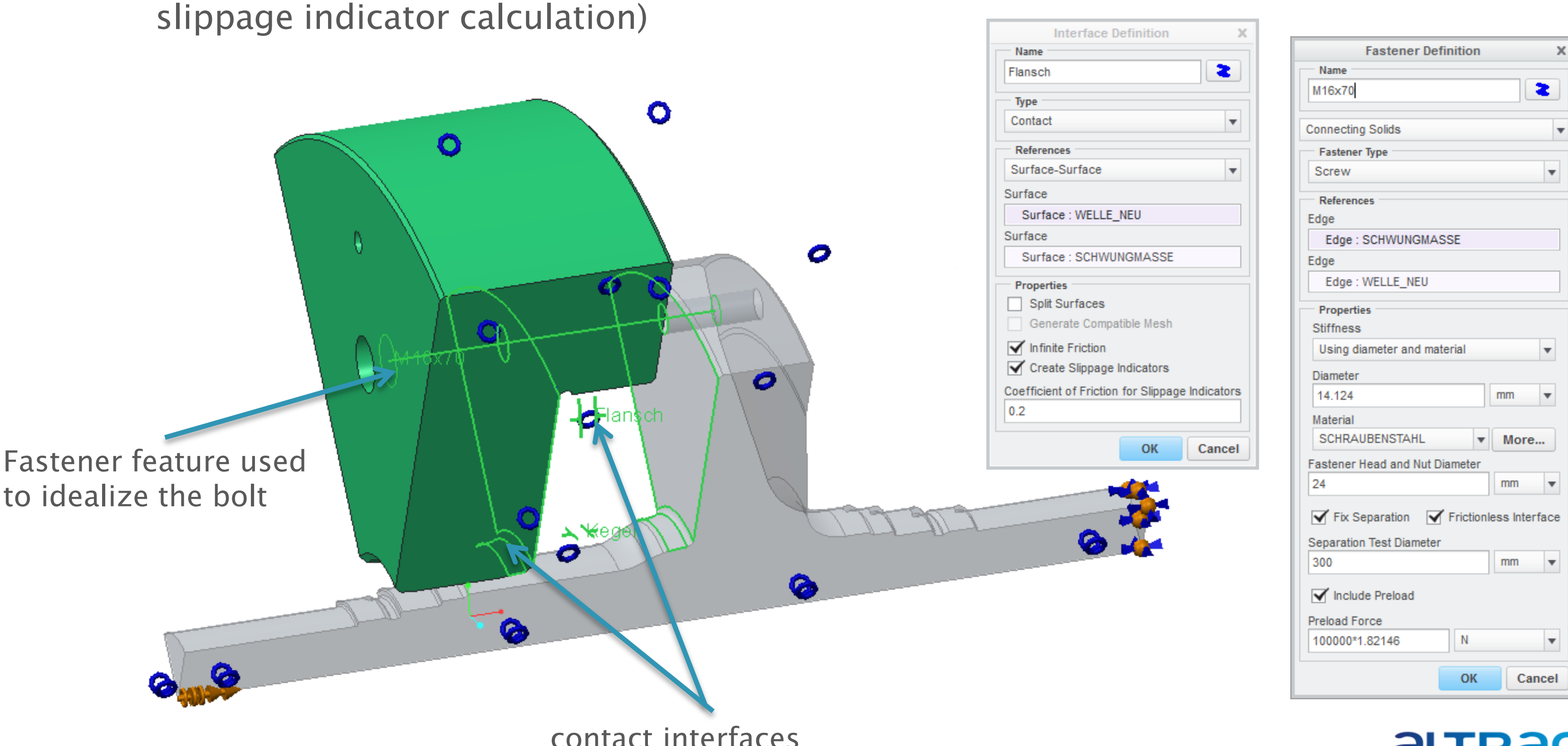


altran

2. Flywheel with a bolted conical hub-shaft-connection

2.2 Simulation model

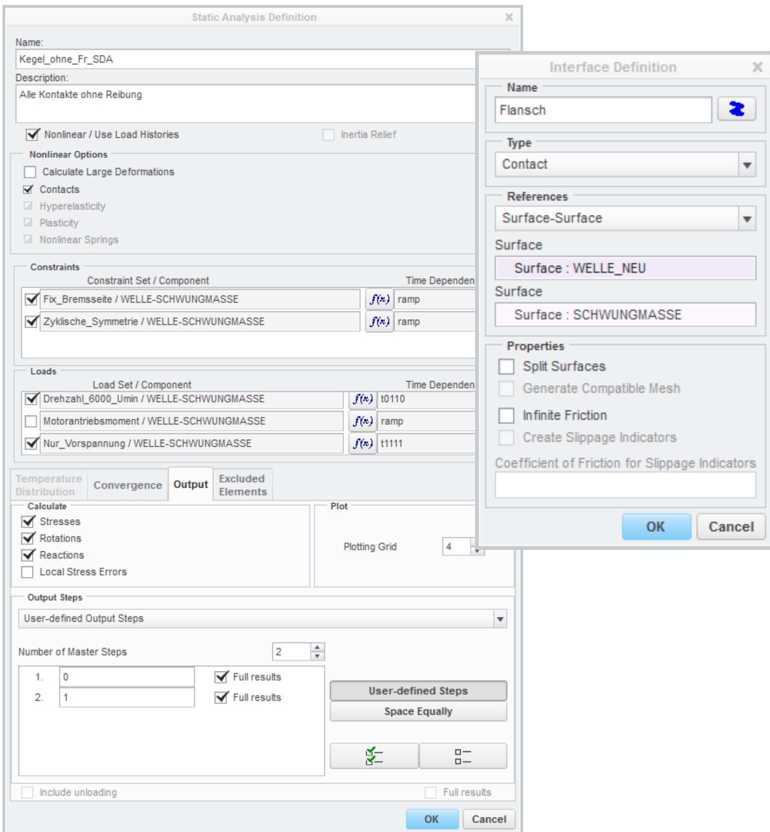
- The model is stripped down to a 60°-segment, to take advantage of the Simulate idealized fastener feature (with a half volume bolt, a 30° segment could be realized)
- First, the model was analyzed in Creo 2 with the friction-free SDA contact model, since this still works pretty robust
- In a second run, the flange interface was set to infinite friction (with μ=0.2 for slippage indicator calculation)



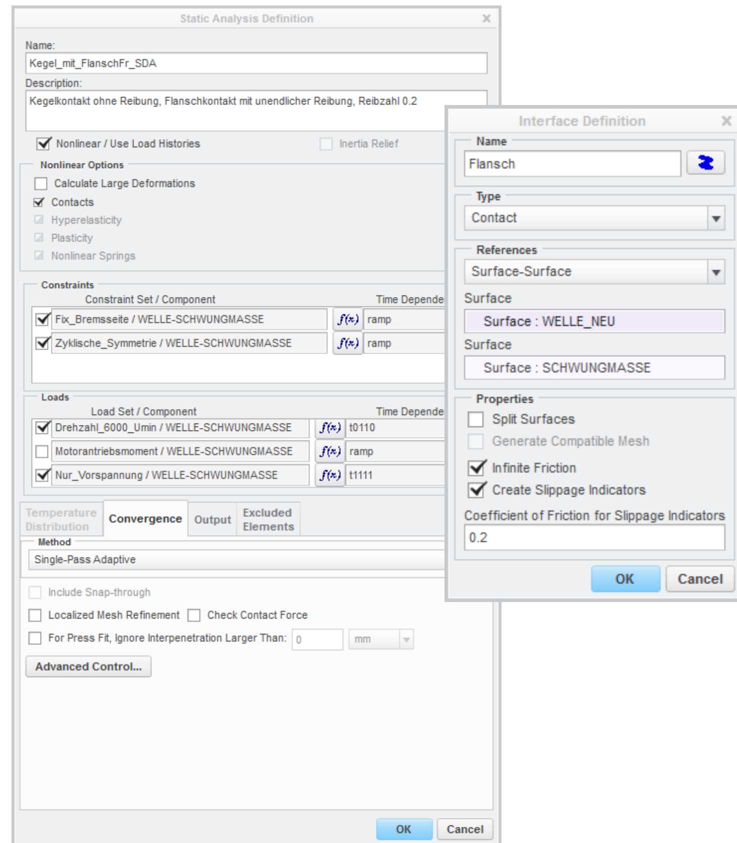
2. Flywheel with a bolted conical hub-shaft-connection

2.3 Analysis with the SDA contact model in Creo 2.0

- Analysis set up in SPA with two load cases:
  Step 1: Only preload – Step 2: additional rotational speed 6000 rpm
- No large displacements necessary to invoke, only SDA



Analysis without friction



Analysis with infinite friction

2. Flywheel with a bolted conical hub-shaft-connection

2.3 Analysis with the SDA contact model in Creo 2.0

- Frictionless contact analysis performs perfectly and needs just 13 minutes to complete

```
Analysis "Kegel_ohne_Fr_SDA" Completed  (13:09:10)

-----------------------------------------------------

Memory and Disk Usage:

   Machine Type: Windows 7 64 Service Pack 1
   RAM Allocation for Solver (megabytes): 4096.0

   Total Elapsed Time (seconds): 761.70
   Total CPU Time     (seconds): 3751.90
   Maximum Memory Usage (kilobytes): 9243745
   Working Directory Disk Usage (kilobytes): 311500
```

- Infinite friction contact analysis needs significantly more iterations to converge and therefore longer (36 minutes), but results are still fine

```
Analysis "Kegel_mit_FlanschFr_SDA" Completed  (13:58:53)

-----------------------------------------------------

Memory and Disk Usage:

   Machine Type: Windows 7 64 Service Pack 1
   RAM Allocation for Solver (megabytes): 4096.0

   Total Elapsed Time (seconds): 2139.89
   Total CPU Time     (seconds): 10783.71
   Maximum Memory Usage (kilobytes): 9238369
   Working Directory Disk Usage (kilobytes): 318671

   Results Directory Size (kilobytes):
   128046 .\Kegel_mit_FlanschFr_SDA
```

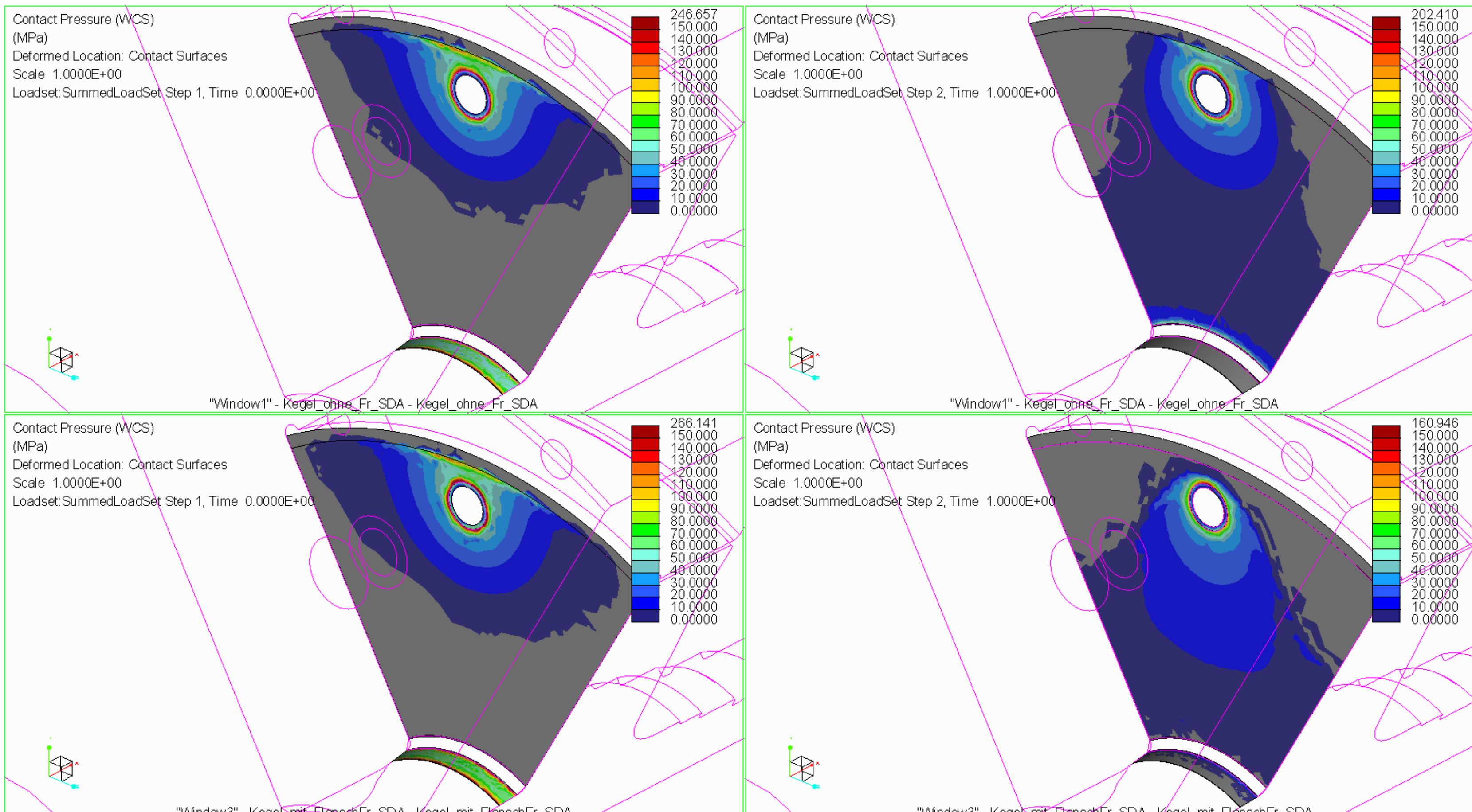
2. Flywheel with a bolted conical hub-shaft-connection

2.3 Analysis with the SDA contact model in Creo 2.0

- Contact pressure results [MPa] / friction-free and infinite friction in comparison

Top: Friction free

Bottom: Infinite friction



Left: Just preloaded

Right: Preload and centrifugal force

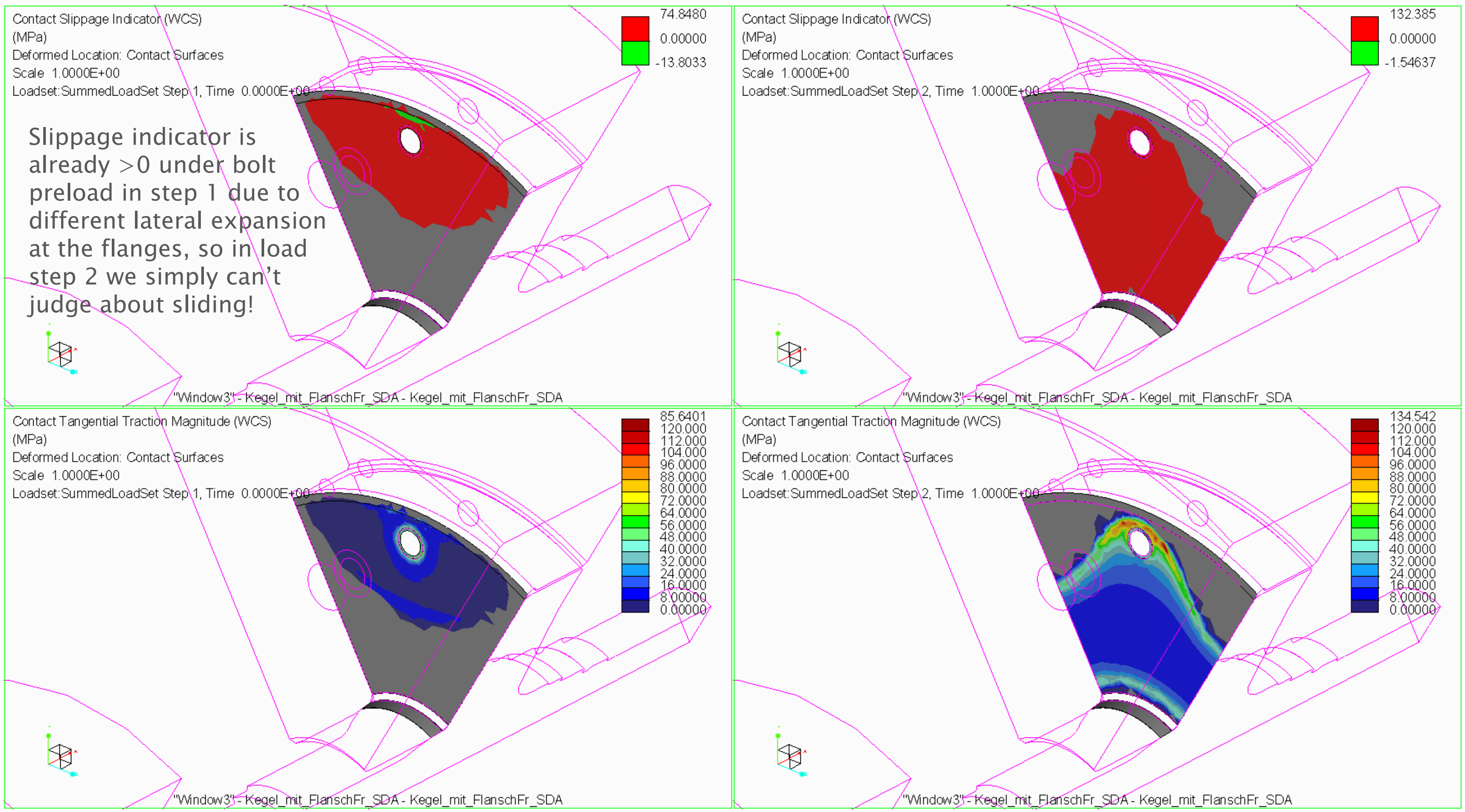
2. Flywheel with a bolted conical hub-shaft-connection

2.3 Analysis with the SDA contact model in Creo 2.0

- Slippage indicator and tangential traction results of the infinite friction contact model:

Contact slippage indicator

Contact tangential traction [MPa]



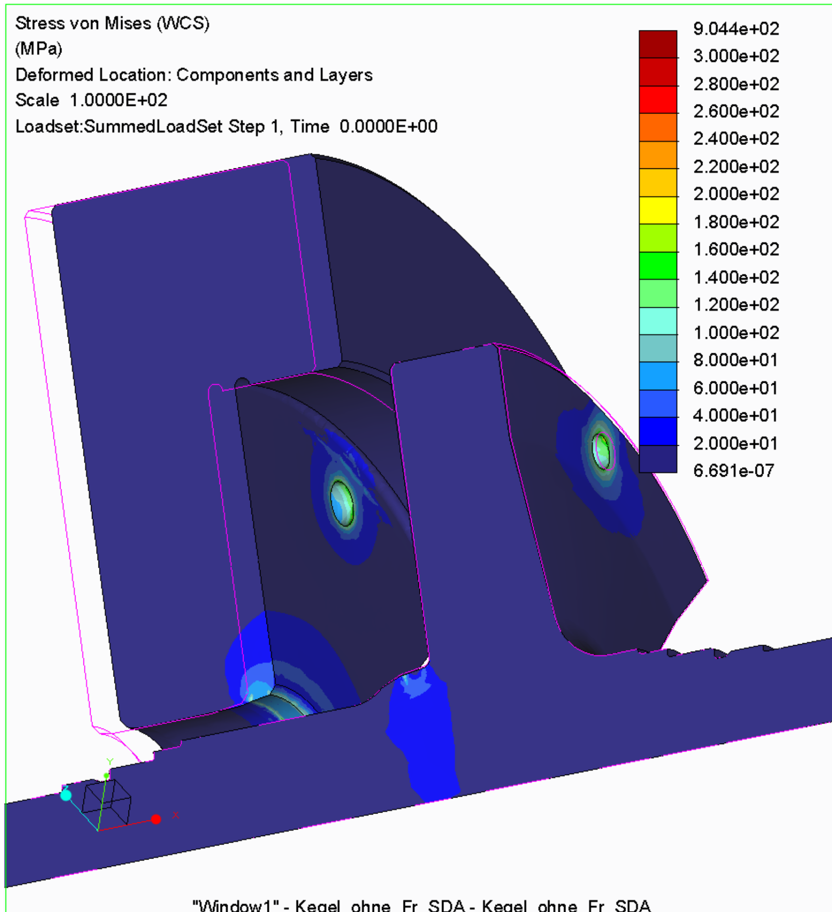
Left: Just preloaded

Right: Preload and centrifugal force

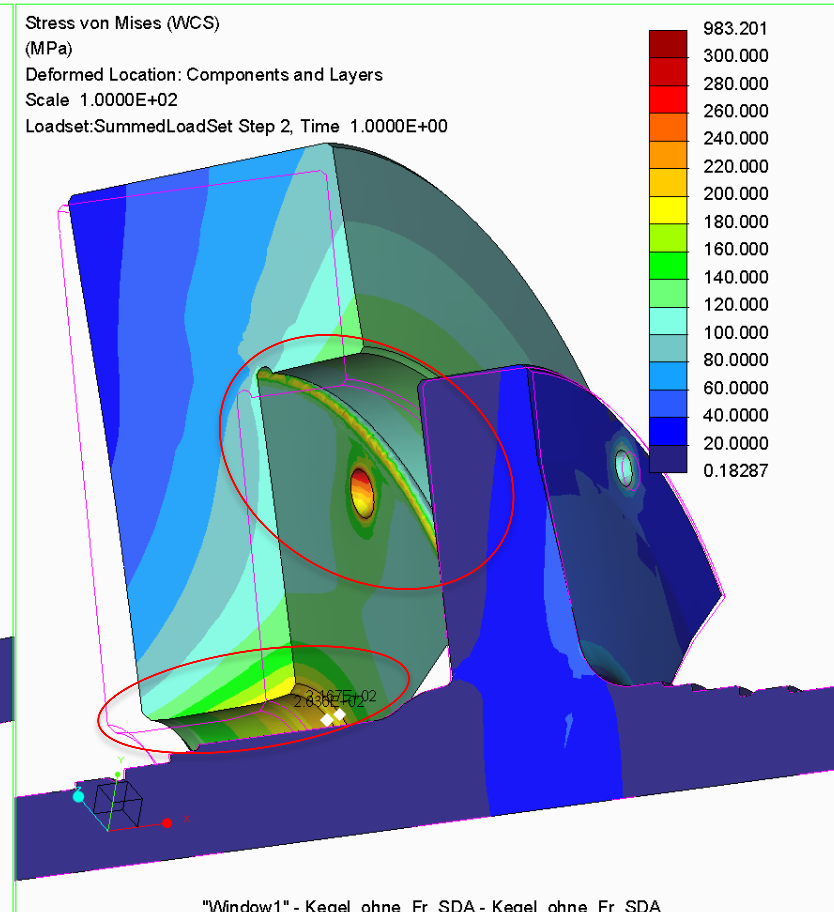
2. Flywheel with a bolted conical hub-shaft-connection

2.3 Analysis with the SDA contact model in Creo 2.0

- Friction free contact model – von Mises stress results [MPa] – disp. scale 100:1



Left: Just preloaded

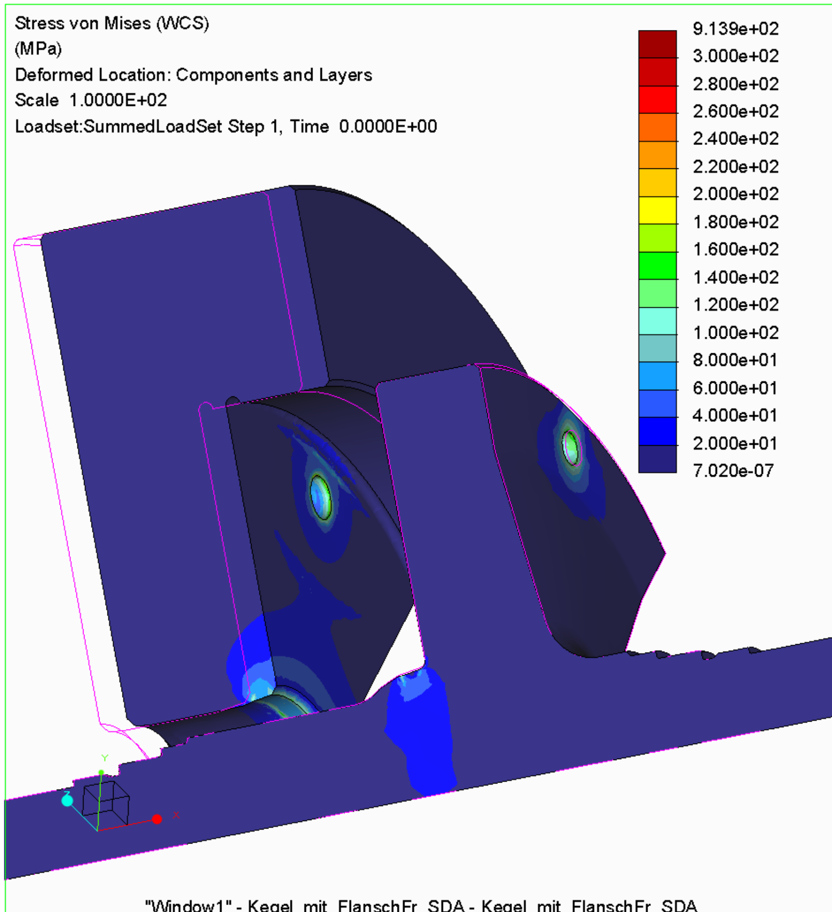


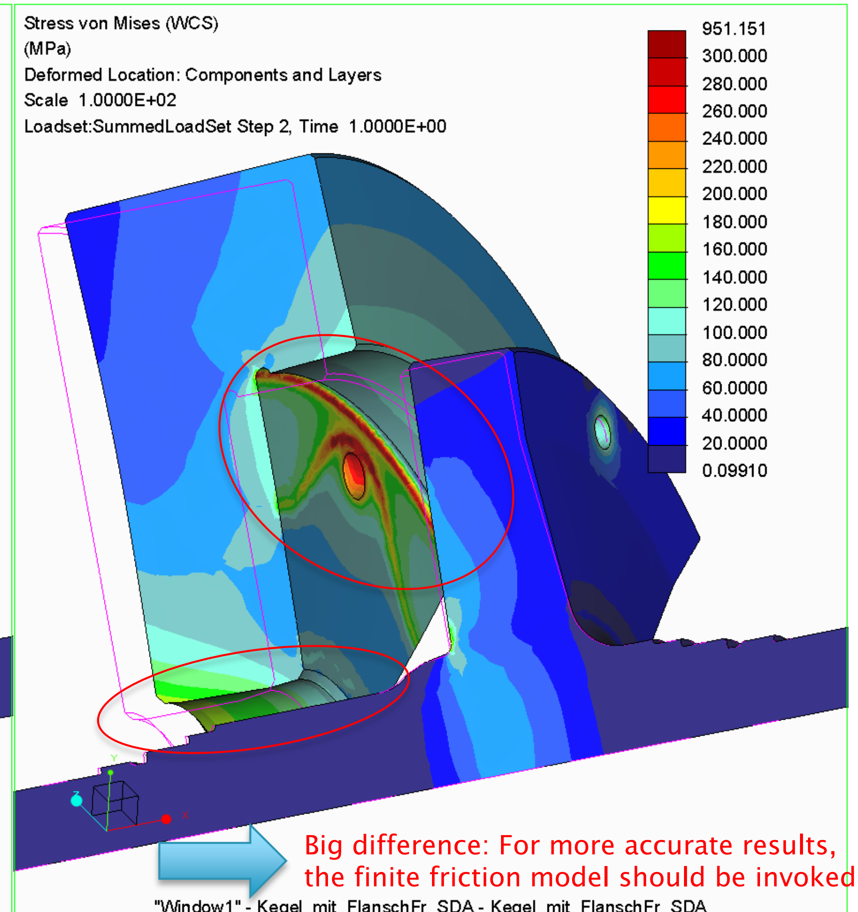
Right: Preload and centrifugal force

2. Flywheel with a bolted conical hub-shaft-connection

2.3 Analysis with the SDA contact model in Creo 2.0

- Infinite friction contact model – von Mises stress results [MPa] – disp. scale 100:1





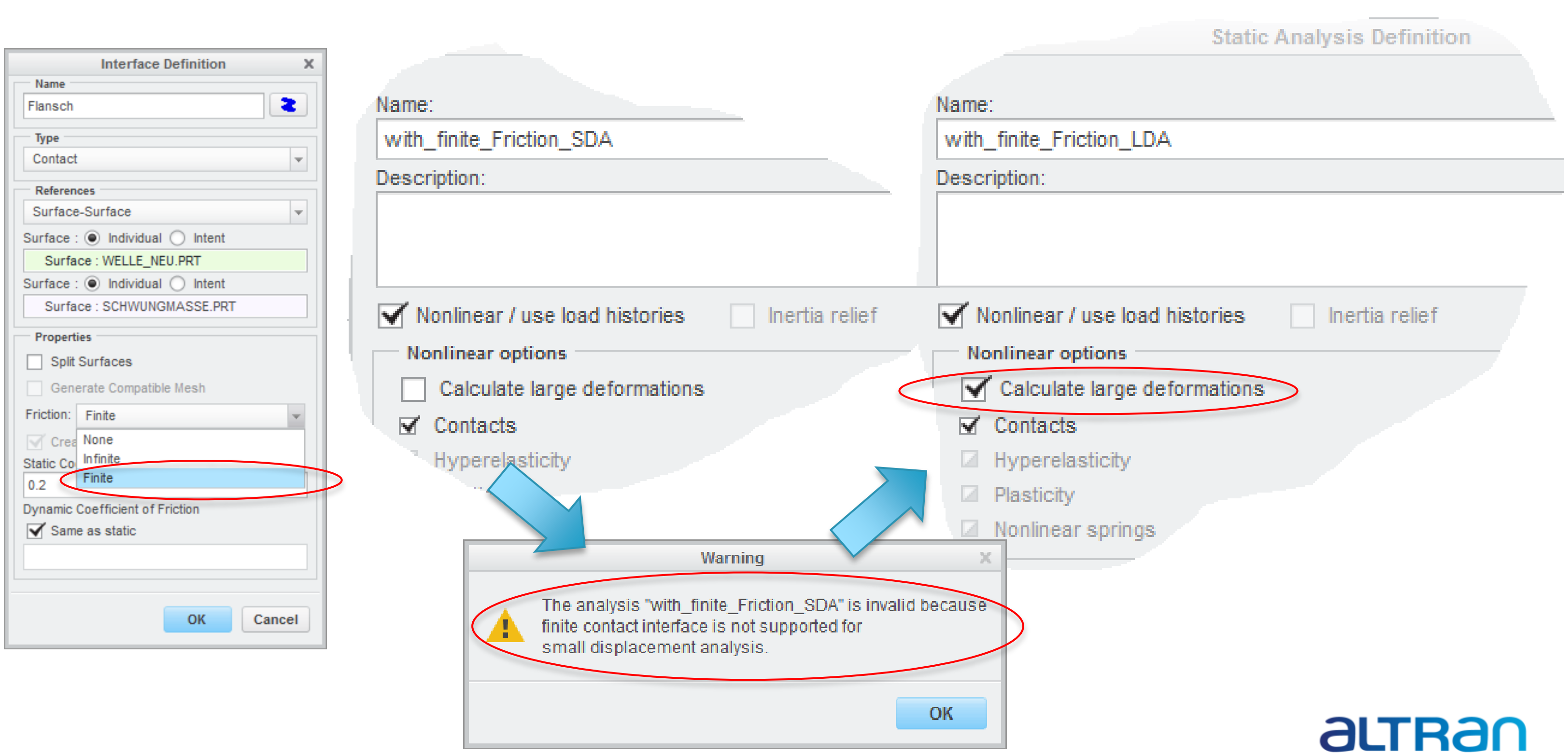
Left: Just preloaded

Right: Preload and centrifugal force

2. Flywheel with a bolted conical hub-shaft-connection

2.4 Transferring the infinite friction model to a finite friction one

- If the user now wants to define a finite friction contact model with help of the model used in the previous chapter, he first has to
  - Redefine the contact interface
  - Switch on “Large Displacements” on the analysis form sheet, since finite friction unfortunately does not support the much simpler SDA theory, which would have been absolutely sufficient for this problem (other FEM codes, like e.g. MARC or ABAQUS, support SDA for finite friction contact since many years)

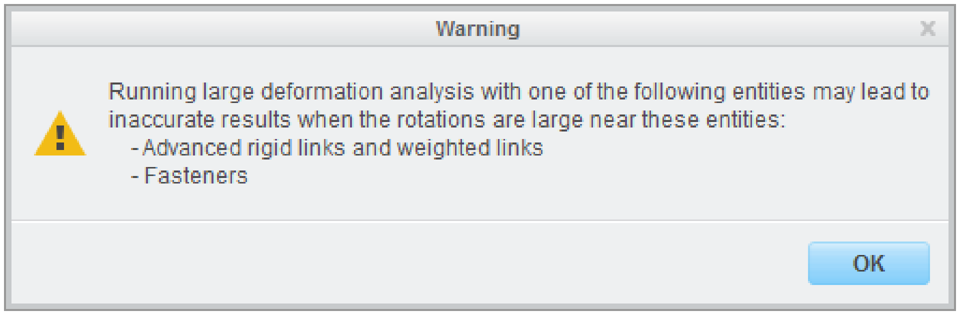


altran

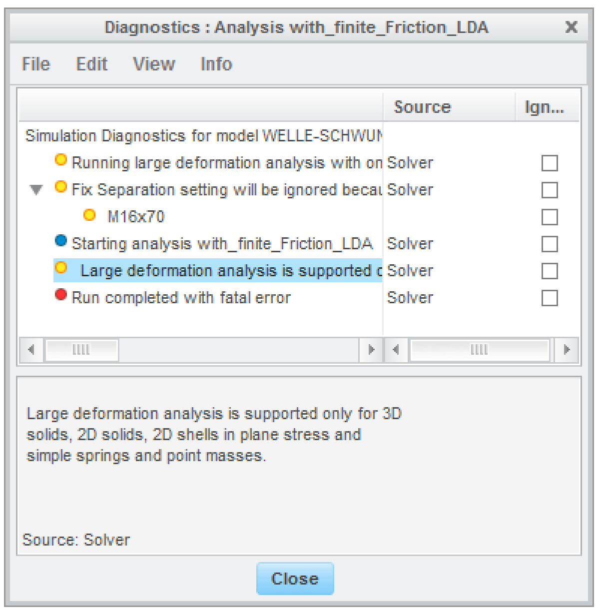
2. Flywheel with a bolted conical hub-shaft-connection

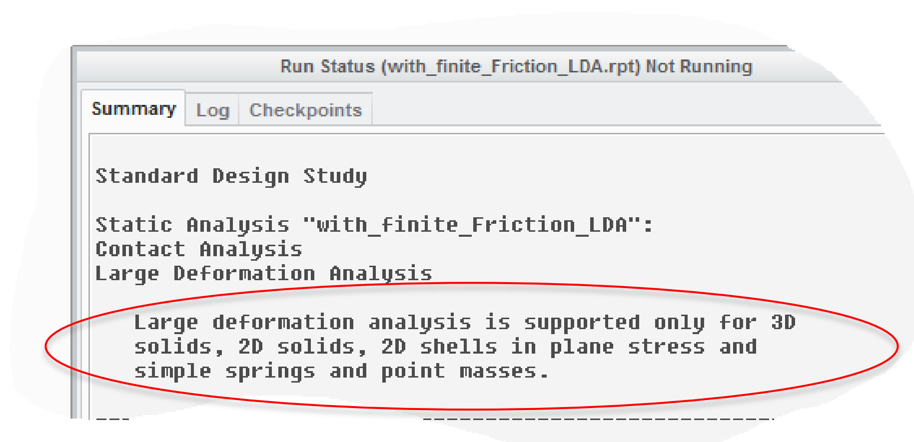
2.4 Transferring the infinite friction model to a finite friction one

- Now, after closing the analysis definition dialogue, the next warning appears:



- The user could ignore this warning, since rotations are very small everywhere for this problem type, but after starting the finite friction analysis an engine error appears:

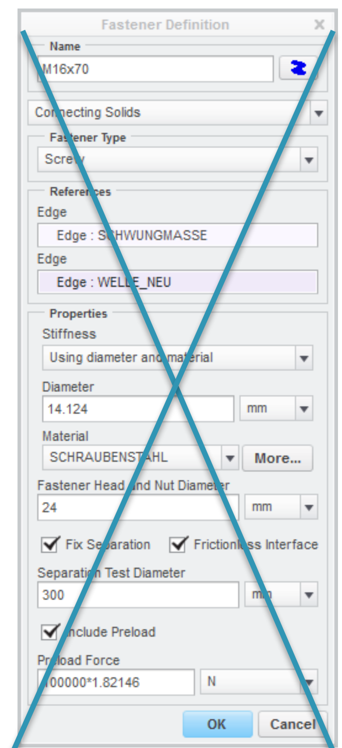


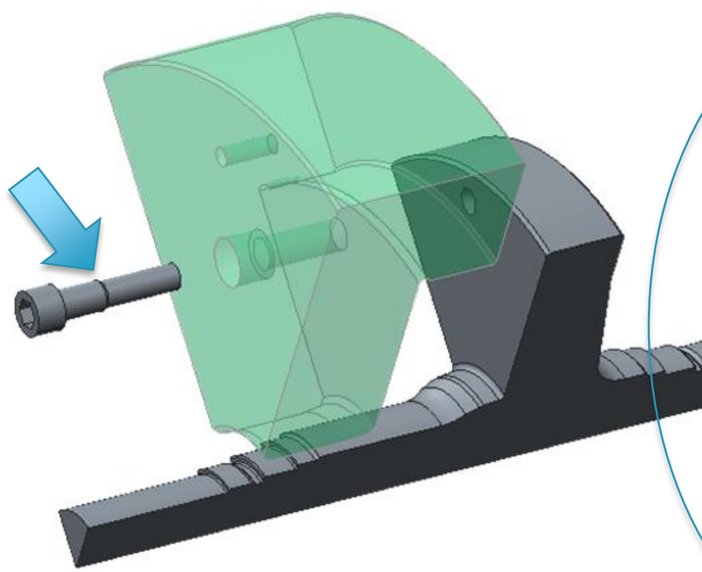


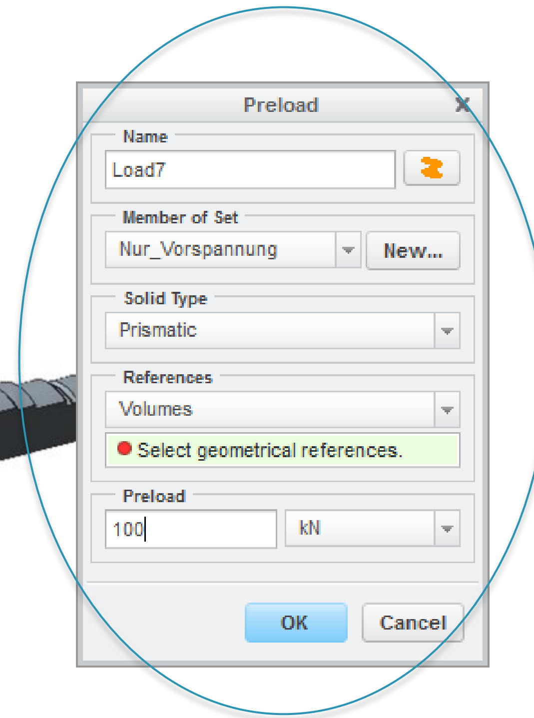
2. Flywheel with a bolted conical hub-shaft-connection

2.4 Transferring the infinite friction model to a finite friction one

- Obviously, what makes the trouble is the used fastener feature, which uses an advanced spring to idealize the bolt, see [8] – but LDA only supports simple springs!
- As a consequence, the user now has to replace the highly idealized fastener feature by a less idealized bolt made of solid elements (commonly used beams are not possible because they are not supported in LDA either!)
- The first intuitive idea now is to use a very user-friendly preload element [8] to apply a preload to the fastener shaft



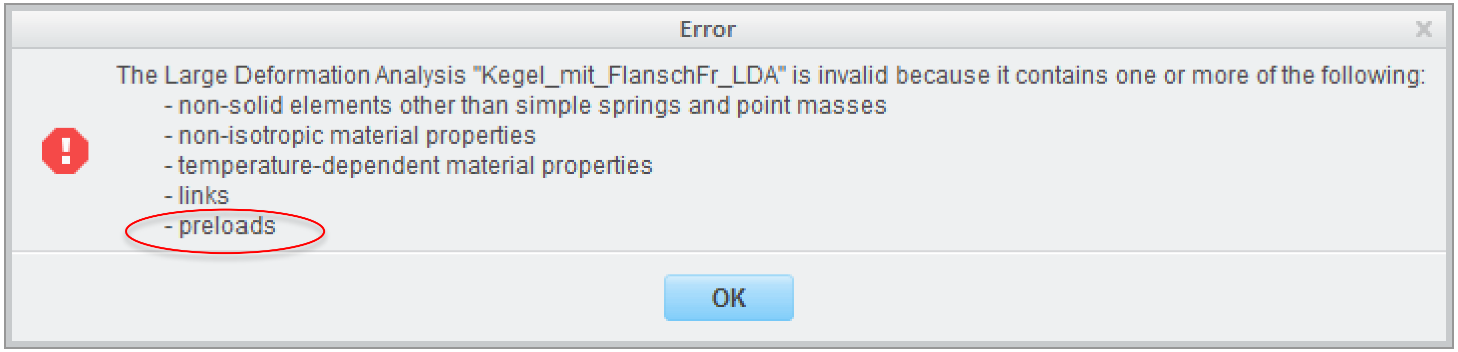




2. Flywheel with a bolted conical hub-shaft-connection

2.4 Transferring the infinite friction model to a finite friction one

- If the user now wants to run the LDA analysis, the next error message appears



- By now at the latest a normal user would give up fully frustrated…

- A Simulate expert will yet try one of the following workarounds:
    1. Apply the bolt preload by an initial interference below the bolt head or at the flange interstice
    2. If no initial interference in the simulation model is preferred, use a thermal load to shrink the bolt shaft

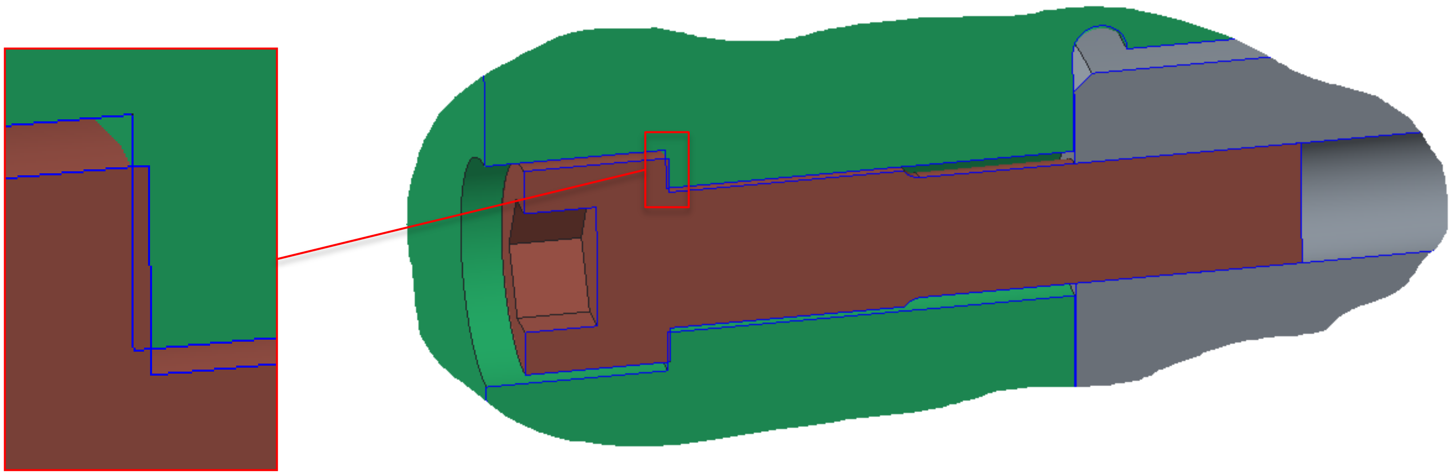
- Both workarounds have certain advantages and disadvantages, as the next slide will explain

altran

2. Flywheel with a bolted conical hub-shaft-connection

2.4 Transferring the infinite friction model to a finite friction one

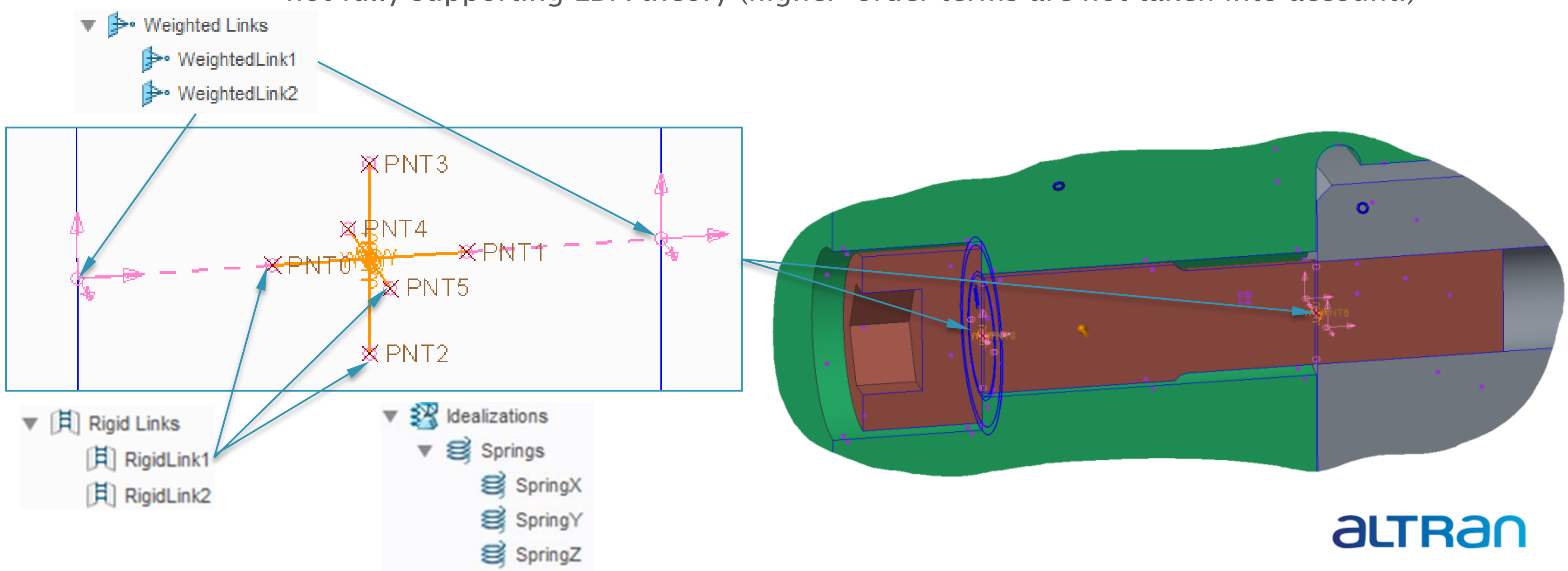
- The disadvantages of option 1 (initial interference) are:
    1. The engine is always forced to do an extra iteration loop at the beginning of the analysis to push the interpenetrating flanks apart
    2. So, you can't control the preload in the load history definition – you can neither switch it on nor off (e.g. to study remaining plastic deformations in the model)
    3. Since you can't apply the initial interference in small steps, usually the finite-friction-LDA fails because of the currently not-robust enough LDA algorithm (whereas SDA contact works pretty fine here!)
    4. Initial interferences often do not work in 2D models (plane stress, plane strain, axial symmetric) since meshing problems may appear at the interference location



2. Flywheel with a bolted conical hub-shaft-connection

2.4 Transferring the infinite friction model to a finite friction one

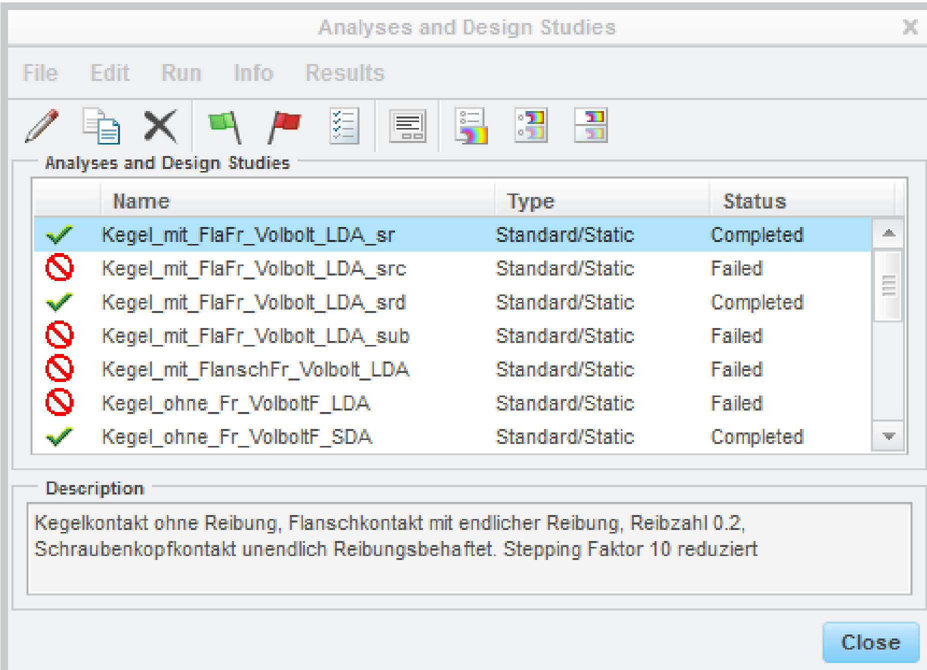
- The disadvantages of option 2 (thermal shrinking without initial interpenetration) are:
  1. Usually, you want a thermal shrinking just in axial direction of the bolt shaft. You can only obtain this by using orthotropic material, where you define a virtual CTE just in axial direction of the shaft. But – orthotropic material is not supported in LDA!
  2. So as workaround in order not to have wrong local stress/stiffness results due to prohibited lateral strains within the bolt, you have to use isotropic material, cut the bolt shaft free and apply the group of “linking elements” shown below – this group uses three simple springs instead of one advanced spring (which is not supported in LDA either)
  3. Since you finally need two weighted links to connect the three orthogonal springs to the neighboring volumes, rotations still must stay small here since weighted links are not fully supporting LDA theory (higher-order terms are not taken into account!)



2. Flywheel with a bolted conical hub-shaft-connection

2.5 Running the 3D flywheel segment as finite friction contact analysis

- Several attempts with different model setups, analysis settings etc. have been tried, but not even one with success



- Either the analyses failed with fatal errors, or they ran until completion, but delivered wrong or very inaccurate results
- To show all model setups and errors in detail would by far go beyond the scope of this presentation, but some typical issues are shown on the next slides

2. Flywheel with a bolted conical hub-shaft-connection

2.5 Running the 3D flywheel segment as finite friction contact analysis

- Often the solution algorithm accepts much too high residua, leading to wrong results, see below (remember the residual norm should be < 1e-12)

```
Iteration     Residual norm   contact Area
------------- ------------- ---------------
            1             1 319.626     Thu Jan 21, 2016 09:48:14
            2   0.000228132 283.979     Thu Jan 21, 2016 09:48:32
            3   0.000103389 162.266     Thu Jan 21, 2016 09:48:35
            4   0.000238254 149.103     Thu Jan 21, 2016 09:48:40
            5       28.6114 335.04     Thu Jan 21, 2016 09:48:56
            6             1 335.04     Thu Jan 21, 2016 09:49:13
***Looser residual tolerance accepted because area and force
   converged for all contacts
```

- This was already reported to PTC R&D as SPR 4633631 dated 22-07-2015 (still open)

- Sometimes the automatic load stepping refinement cuts load step size down to values close to zero for whatever reason and a fatal error is reported (see right)

Run Status (Kegel_ohne_Fr_VolboltF_LDA.pas) Not Running

Summary | Log | Checkpoints

```
Iteration     Residual norm   contact Area
------------- ------------- ---------------
            1             1 335.04      Thu Jan 21, 2016 09:53:55
            2   4.70132e+006 586.983     Thu Jan 21, 2016 09:54:12
            3             1 586.983     Thu Jan 21, 2016 09:54:24

Load Factor:          0.5

Load Factor:         0.25

Iteration     Residual norm   contact Area
------------- ------------- ---------------
            1     0.0312425 336.52      Thu Jan 21, 2016 10:07:34
            2             1 336.52      Thu Jan 21, 2016 10:07:35

Load Factor:        0.125

Load Factor:       0.0625

Load Factor:      0.03125

Load Factor:     0.015625

Iteration     Residual norm   contact Area
------------- ------------- ---------------
            1   0.000122041 560.751     Thu Jan 21, 2016 10:20:19

Load Factor:    0.0078125

Iteration     Residual norm   contact Area
------------- ------------- ---------------
            1  3.05103e-005 560.896     Thu Jan 21, 2016 10:28:09
            2             1 560.896     Thu Jan 21, 2016 10:28:24

Load Factor:   0.00390625

Load Factor:   0.00195313

Load Factor:  0.000976563
*** A fatal error has occurred. ***
The nonlinear iteration did not converge for the
time value:  9.76563e-04

   o The model may have a buckling mode with a load factor
     smaller than this value. You may determine the smallest
     (linear) buckling load factor by running a buckling
     analysis.

   o The deformations may be large enough that you must
```

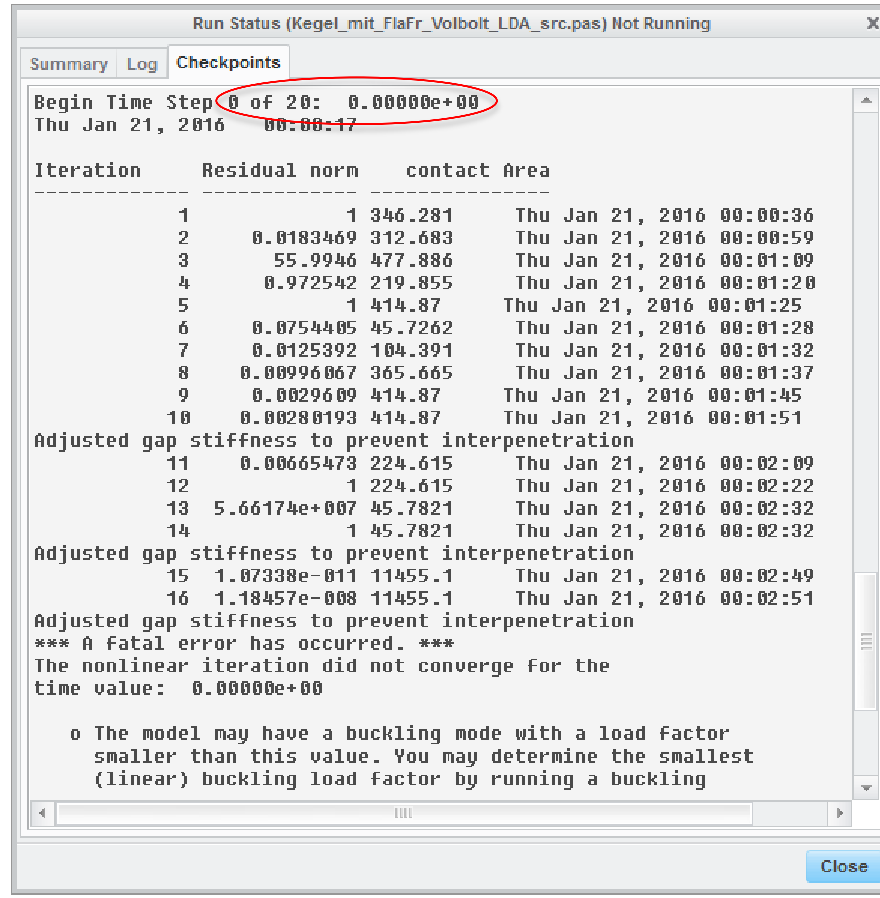
Close

altran

2. Flywheel with a bolted conical hub-shaft-connection

2.5 Running the 3D flywheel segment as finite friction contact analysis

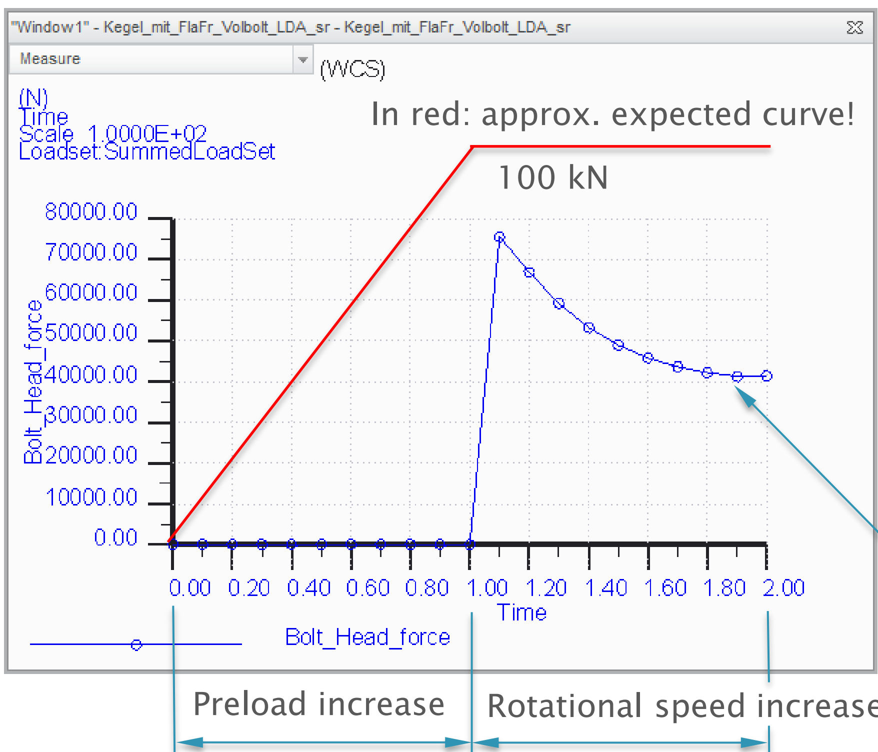
- Even though also models have been tested that used no initial interpene– tration and no external force was applied in load step Zero, the iteration unnecessarily may start at this Zero load step and fail, see right

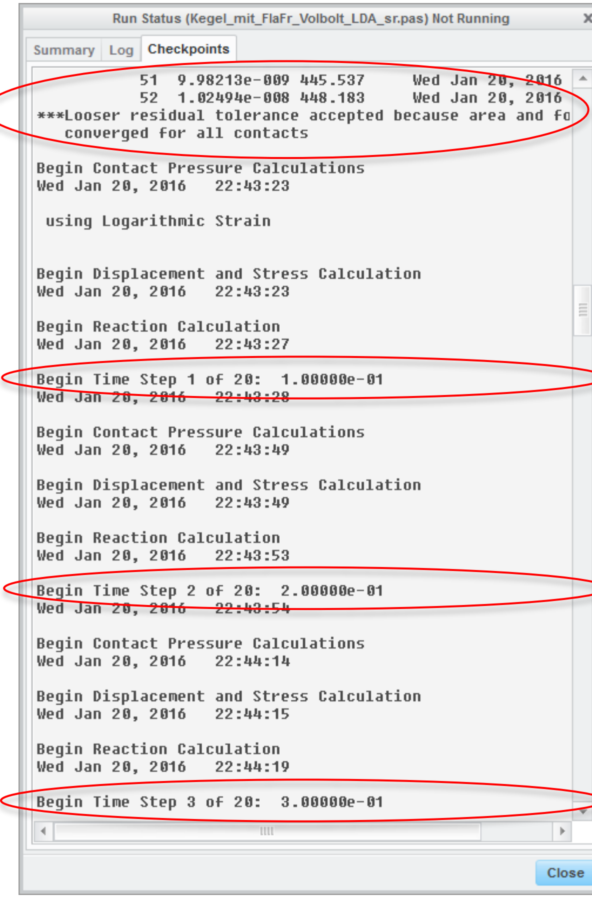


2. Flywheel with a bolted conical hub-shaft-connection

2.5 Running the 3D flywheel segment as finite friction contact analysis

- It may also happen that the algorithm simply does not detect interference at the flange during preload increase, and no iteration takes place (shown right)
- Wrong results will be the consequence, e.g. bolt force measures like shown below



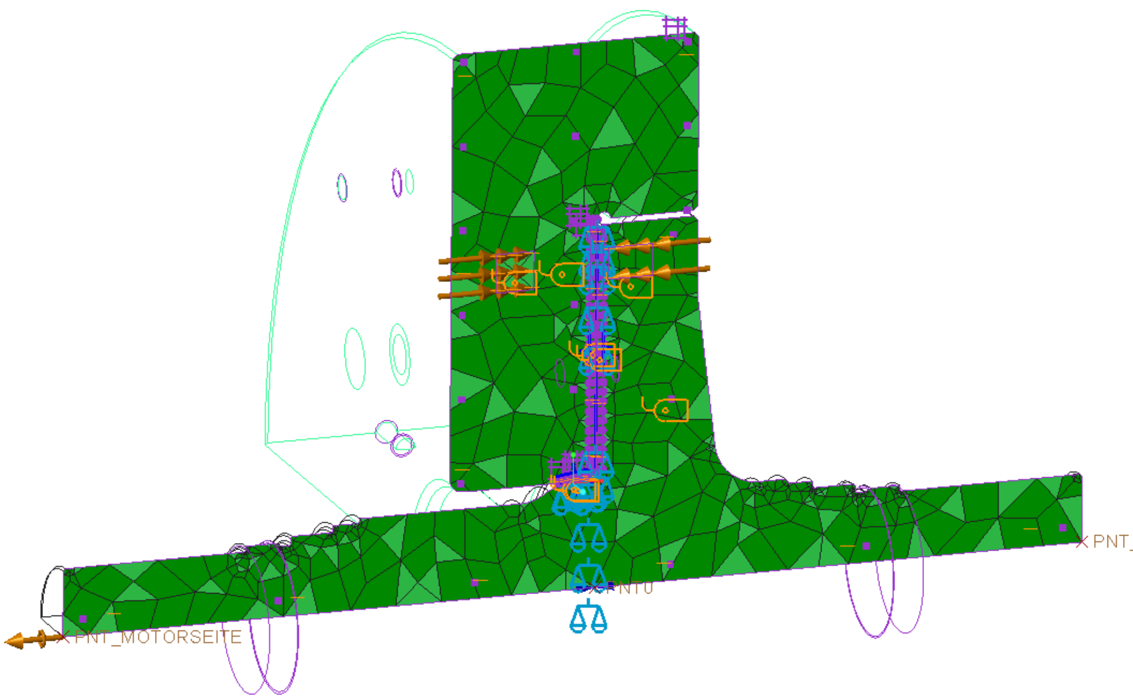


The algorithm may also increase the interpenetration during the load increments at the contact flange, so that there is an unreasonable preload loss

2. Flywheel with a bolted conical hub-shaft-connection

2.6 Running the flywheel as 2D axial symmetric finite friction contact analysis

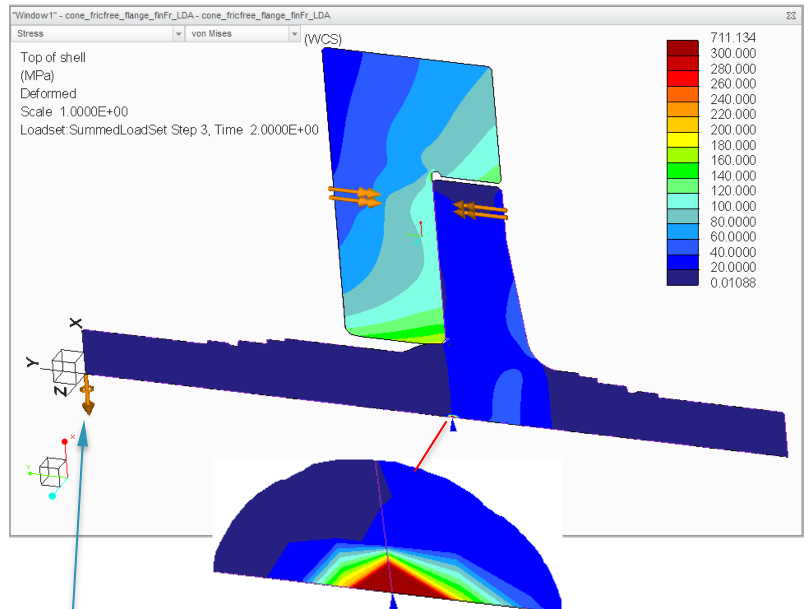
- A last attempt was done to alternatively set up the model as 2D axial symmetric model to obtain at least some approximated results
- This can be done in this special case since the bolt (preload) can be replaced by an equivalent force pair at the bolt circle diameter
- This “cutting away” of the bolt spring stiffness from the complete mechanical system is allowed only since the bolt is only shear and not normal loaded, and the shear spring is very soft compared to the attached flanges
- The obtained 2D model therefore looks like shown below



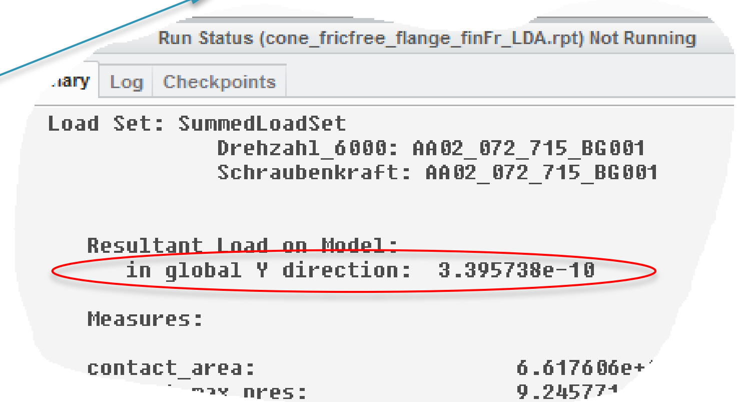
2. Flywheel with a bolted conical hub-shaft-connection

2.6 Running the flywheel as 2D axial symmetric finite friction contact analysis

- Unfortunately, also with this simple model no error-free results with the finite friction model could be obtained
- Furthermore, already for the friction free and infinite friction contact in SDA a couple of wrong measure results were detected
- For the finite friction model, again the contact force measures became totally wrong (some potencies of 10 off)
- Furthermore, for the finite friction analysis a hot spot at a constraint was computed, even though the model is balanced and the report file reports zero resulting force in Y-direction
- This was reported to PTC as Case 12907045 / SPR 5178330



Note the PP erroneously displays load and constraint directions wrong!



Part C: Feedback to PTC

Feedback to PTC

1. Experience regarding the different contact models in Creo 2.0 & 3.0

Contact model:	Friction free	Infinite friction	Finite friction
Experience won with the model	Very good (state Creo 2.0 M200); works quick, robust and reliable in most cases	Contains a significant risk to obtain erroneous or at least inaccurate results (Creo 2.0 M200)	Absolutely unsatisfying and unreliable, wasted time even to test (state Creo 3.0 M080)
Success rate (estimated value from project application experience)	>95 %, at least when used with SDA and linear material	60–70 %	0 %
Typical error examples/ problems observed	May underestimate Hertz contact pressure/contact stress with default settings	1. Often shows poor convergence / many iterations necessary (very slow) 2. May typically compute too much penetration and as consequence e.g. too low bolts loads at interpenetrating flanges	1. Fails with fatal error for any reason (stability issues, cuts down load step size until failure,…) 2. If the analysis completes, usually inaccurate or wrong results are obtained, often with too much interpenetration
Possible solutions	usually a refined mesh and reducing contact penetration helps	1. increase allowed number of contact iterations >200 2. Unfortunately, this often cannot be fixed by reducing contact penetration, then try other options shown in this presentation	Non – PTC R&D: Rework and fix the code!

Feedback to PTC

1. Experience regarding the different contact models in Creo 2.0 and 3.0

Most important issues to fix:

- Urgently completely rework the finite friction contact model, in this quality state it is practically unusable (since July 2014, when Creo 3.0 F000 came out)
- Improve stability & reliability and increase speed of the infinite friction model
- Change the spring force based analysis of the contact load measure in Creo 3.0 back to the element normal stress based approach used until Creo 2.0, to give users again the opportunity to do quality assurance for contact stress and pressure results
- In addition to the code problems, improve the program documentation and deliver more detailed information about engine and “hidden options” functionality, it’s a lot of work or even impossible to try this out by “reverse engineering”!

Most important enhancements:

- Implement the finite friction contact model asap for small displacement analysis (SDA)
- Remove all the code’s LDA limitations, so that finite friction models can also take benefit of e.g. shell and beam idealizations, advanced springs, fastener features in complete system analyses!

General remarks:

- In general, it was pretty difficult to obtain *any* finite friction contact model example running until completion at all! Until today, we never got a finite friction customer project model successfully analyzed, even though we try since Creo 3.0 F000 (2014)
- As consequence, Altran had to discontinue offering to solve finite friction contact problems using Creo Simulate!

Feedback to PTC

2. General experience with Creo Simulate at Altran

General Situation

- Altran has currently approx. 50 open SPR regarding Creo Simulate, many with high priority
- 60-70 % of these SPRs are engine related
- Many of them are 1-2 years old, some even older, for example:
  - SPR 2868682: Incorrectly working nonlinear stability analysis (missed snap through events), opened 5-Sep-2013, fix planned for Creo 2.0 M220
  - SPR 2875703: Wrong results display for dynamic frequency analysis with force excitation and phase differences between the exciting forces (5-Apr.-2015, but other SPRs for this issue exist since 2013), planned to be fixed as enhancement (!) for Creo 5.0
- Nearly none of all these issues is reported in the PTC Technical Support eNews & Alerts for Simulate, even though several issues create wrong results and are not model specific
- Because of all the trouble observed, Altran uses mostly the more proven Creo 2.0 release since we are afraid of finding many more issues in the obviously insufficiently QA-tested Creo Simulate 3.0 release and loose still more project time and money

altran

Feedback to PTC

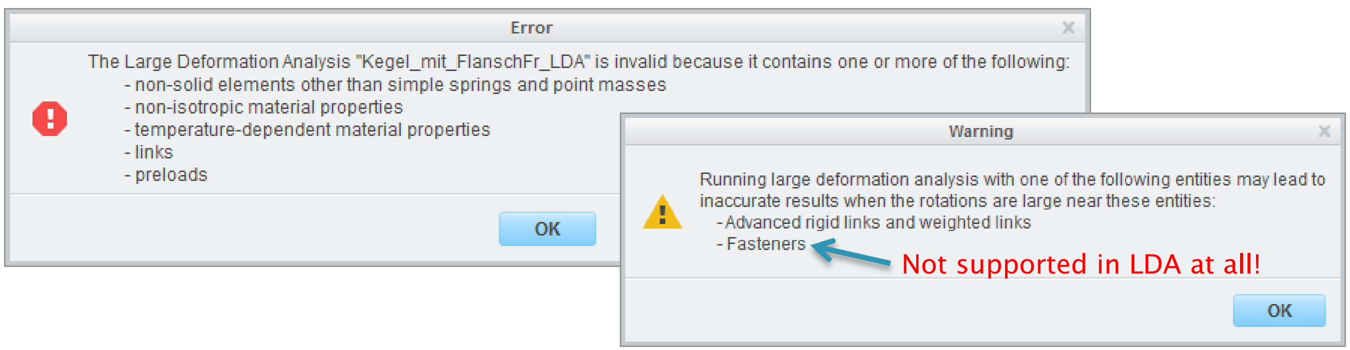
3. Altran's plans for the future regarding Creo Simulate

- Despite dozens of found issues and all related trouble, *Altran currently does not plan to replace Simulate by another Simulation code*, since yet some things are still unsurpassed, like the seamless CAD integration or the well structured and very fast to use Creo UI – but competitors work hard to close this gap!
- We also value the possibility to prepare huge and complex Creo Parametric CAD-assemblies for linear static and dynamic system analysis and take advantage of the associativity between all the simulation features and the CAD geometry, which allows an extremely quick iterative "manual" design optimization if applied by experienced experts. We would like to do this also with LDA problems!
- Also the integrated parameter optimizer with the option to perform global and local sensitivity studies is still used by Altran with great success for notch stress minimization, see [6] and [7] (SAXSIM presentations of 2014 and 2015)
- In the linear domain, mostly the code still works fine, with a couple of exceptions, like
    - SPR 2847768: Wrong von Mises stress hot spots in random response analysis
    - SPR 2875703: Wrong results display for dynamic frequency analysis with force excitation and phase differences between the exciting forces
    - SPR 2873817/2258467: Wrong non-symmetric results for a symmetric simple cone under internal pressure
    - SPR 4948841/2848377: Wrong beam stress results
- Anyway, especially the nonlinear functionality of the engine currently is a huge construction site and urgently has to be quality improved and further developed
- We expect from PTC that all issues are fixed in an acceptable time span, so not within further years, but months!

Feedback to PTC

4. Comments to PTCs planned enhancements for Creo Simulate 4.0

- The news announced by PTC for Creo Simulate 4.0 just address usability, <u>not even one engine functionality enhancement or even robustness increase</u> is planned
- Of course it is nice to have these usability enhancements, but, like this presentation shows, usability is not what we miss, since it is still unsurpassed
- First we need the engine to become robust and reliable again, and we need to close the bunch of existing functionality gaps regarding large deformation analysis (LDA), so that finally we can apply LDA (so e.g. the finite friction contact model) without at least the following limitations (like other codes do since many, many years!):



- Altran has already provided a long list of enhancements to PTC after the engine group in San Jose was laid off in October 2013
- Unfortunately, non of these enhancements has been taken into account until today

Feedback to PTC

4. Comments to PTCs planned enhancements for Creo Simulate 4.0

It now appears that laying off the former engine group in San Jose with a couple of very experiences engine coders in October 2013 lead to a big loss in knowledge and brain power to properly maintain and further develop the code, see

- https://www.ptcusercommunity.com/message/245980#245980
  "The end of Creo Simulate?"

ptc-5259023 13.06.2014 13:58 (als Antwort auf: mfischer)

Re: The end of Creo Simulate ?

An important correction: that core development team "from those early years of Pro/MECHANICA at PTC" was laid off in October. The "center of excellence for Simulation development" in India is now staffed with just several developers who mainly played a supporting role during last few years.

Good luck to you all.

For more information, Google "The end of Creo Simulate" and see the thread on eng-tips.com

Aktionen Gefällt mir (0)

- https://www.ptcusercommunity.com/message/429600#429600
  "What ever happened to Tad Doxsee?"

ChrisKaswer 19.11.2015 06:13 (als Antwort auf: mlindqvist)

Re: What ever happend to Tad Doxsee?

Yes - the entire team that had been in San Jose CA was eliminated - nobody remained with the company ... HUGE loss. There were a few people who had been there from the very beginning (Rasna Mechanica days!). All development that had been done there was moved to a PTC location in India, where there had already been a chunk of work for Simulate being conducted over the past 10+ years.

my 2 cents

Aktionen Gefällt mir (1)

altran

Feedback to PTC

4. Comments to PTCs planned enhancements for Creo Simulate 4.0

http://www.eng-tips.com/viewthread.cfm?qid=356756:

creosimulateuser (Mechanical) **(OP)** 31 Jan 14 01:29

That's exactly what I was talking about. The most experienced developers and managers were all located in the US. There also were a few developers in India who played supporting role, and now they are tasked with supporting and advancing entire product line. One can imagine what can come out of this... especially in terms of product quality.

Of course PTC will try to convince you that nothing changed, they need your maintenance money.

PTCFischer (Structural) 31 Jan 14 14:37

Hi Shaun8567,

There has been a lot of gossip around the departure of the San Jose Simulation development team. Let me try to address your concerns....

In an effort to optimize the product development of Creo Simulate we looked to create a center of excellence for Simulation development. The chosen location was India and thus we shut down the San Jose office. India has and will remain the main development center for Creo Simulate and the rest of our simulation products (MDX/MDO and BMX). The team in India (15 people focused on Creo Simulate) has been actively developing Creo Simulate for several years (most 10+ years), and have a deep level of understanding of the product. This team remains focused and are committed to the success of Creo Simulate.

While the team in San Jose were focused on specific details of Simulate - Mesher and Solver Engine, the team in India worked hand in hand with them to introduce functionality, resolve issue and support the product.

Last week we presented to the Simulate TC as part of the midyear TC event at PTC HQ. Along with our development lead, I presented the new functionality coming in Creo 3.0, as well as, our thoughts for Creo 4.0. The TC members were very encouraged with the direction and what is on the horizon.

As I stated to creosimulateuser, if you have questions pertaining to this discussion, I welcome your emails and comments. You can email me directly at mfischer@ptc.com.

Regards,
Mark

shaun8567 (Mechanical) 5 Feb 14 18:33

"In an effort to optimize the product development of Creo Simulate we looked to create a center of excellence for Simulation development. The chosen location was India and thus we shut down the San Jose office."

So, to put to plainly, the US development team was outsourced to India to reduce operations cost? That's the only reasoning I can come up with.

I had the privilege about a year ago to go to the San Jose facility and speak with some of the developers (like Christos, Tad, and Eduardo), so I'm a little worried what kind of impact this will have on the software due to the loss in brain trust (if I recall correctly, they all have PhD's from very prestigious universities). I'm also a little worries how this will effect the robustness/stability of the software, and whether there is/will be an impact on service tickets. I know that, for example, between WF5 and Creo 2.0 there seems to be an issue with the solver when doing a LDA with arc-length control active. The model solves in WF5, but fails in 2.0 (I have already submitted a ticket, just haven't gotten a response yet).

Author's comment: The solver engine is the heart of each Finite Element code, not a specific detail! Removing qualified personnel from this pretty important part of the product means degrading code quality, loosing competitive capacity to competitors codes, and displease customers. PTC should react here immediately!

Feedback to PTC

5. Experience of other customers

Regarding finite friction contact, follow e.g.

- https://www.ptcusercommunity.com/message/411103#411103:

auijttewaal 06.07.2015 07:05 (als Antwort auf: 346gnu)

Re: Finite friction blocks

Hi All,

PTC has acknowledged that the tangential forces are NOT correct; SPR 4571852 with HIGH PRIORITY, for those who can view it.

I'm not sure about the rest of the results;

-You're 'average slippage' measure is (very) negative, so telling you there is NO SLIPPAGE (seems correct)

-The displacement you show is only in the outline of the block, not in the centre, so is this slippage or just deformation of the block as a result of the forces?

Furthermore, it is clear that to get (probably) correct displacement results with finate friction (in the current release) you MUST have about 11 time-steps and a very fine mesh. With standard analyses settings the study will report large displacement but in my experience it does not report that convergence is not obtained. However, the fact that it jumps in a SPA from poly 3 to 9 does indicate it.

Aktionen | Gefällt mir (0)

auijttewaal 22.07.2015 06:22 (als Antwort auf: 346gnu)

Re: Finite friction blocks

Dear Charles,

New work around provided by PTC R&D;

In the current release of Creo 3 (m040) in order to get correct results for Interface_force with finite friction you need to set the following settings;

- Contact should be surface-surface (not component-component)
- Use study type Quick Check (SPA is currently not correct)
- Ensure you have at least 10-11 time steps

I've tested it and my Interface_force is now correct! Try it, I would say...

I'm still in discussion with R&D about the tang_traction and the max_tang_force results...

Feedback to PTC

5. Experience of other customers

- Under https://www.ptcusercommunity.com/thread/59938, you find for example:

sdensberger 08.02.2016 13:03 (als Antwort auf: jguelcher-2)

Re: Non-symmetric results for symmetric model and load?

I genuinely wonder if the R&D team has a QA process in place, because this is something that a suite of benchmark problems should have caught. Honestly, this is unacceptable for a solver; this is basic linear-elastic mechanics here.

Aktionen — Gefällt mir (1)

unickque 08.02.2016 23:43 (als Antwort auf: sdensberger)

Re: Non-symmetric results for symmetric model and load?

Making a mistake is not so bad, we are all human and we all make mistakes.

How I always judge people and companies is how they deal with the mistakes they make.
In the case of PTC, they are not doing very well so far. It's been almost a year and still no news on a fix... that is very bad for a serious issue such as this.

Aktionen — Gefällt mir (1)

sdunker 10.02.2016 05:18 (als Antwort auf: sdensberger)

Re: Non-symmetric results for symmetric model and load?

This problem needs to be addressed by Mark Fisher, because is the Product Manager for Simulate.

Aktionen — Gefällt mir (0)

altran

Feedback to PTC

5. Experience of other customers

- https://www.ptcusercommunity.com/message/433384#433384:

mlindqvist 09.12.2015 05:29 (als Antwort auf: ehaenen)

Re: Enforced displacement, tangential direction, more than 1 revolution

I found an SPR some time ago, for a bug that cost me 2 days of searching for modeling errors. PTC's plan for this particular bug looks like this:

Creo 2.0 No plan to fix
Creo 3.0 No plan to fix
Creo 4.0 No plan to fix

SPR Details - 2848377 / 2217099

PTC recommends using this tool to help diagnose issues being encountered while running PTC software or as a part in determining what value there is in updating to a particular Release or MOR.

Please note that SPR descriptions frequently do not detail enough information to determine a match with your software behavior. Simply stated, different SPRs can have similar symptoms. Conversely, an SPR can exhibit multiple symptoms that may or may not be captured in the description.

SPR information is offered for your limited review and within context. PTC Technical Support recommends weighing the above factors when reviewing the SPR information. In all cases where it applies, refer to the TAN associated with the SPR and contact Technical Support if you need further information.

SPR	2848377 / 2217099
Status	Closed
Severity	High
Created Date	27-MAR-2014
Description	72150 N Load, corresponds to a beam tensile stress of approx. 460 MPa (cross section 157 mm). However, the postprocessor reports beam compression stresses between -168.983 and -636.930 N, which is totally wrong. Yo
Affected Platform	All

Affected Products

Reported Product	Creo Simulate	
Module	Creo Simulate	
Reported Release	Creo 2.0	
Reported Datecode	M100	
Resolution Status		
Release	**Status**	**Datecode**
Creo 2.0	No Plans to Fix	-
Creo 3.0	No Plans to Fix	-
Creo 4.0	No Plans to Fix	-
Product Family	Creo	

Aktionen ▾ — Gefällt mir (1)

sdensberger 28.12.2015 18:14 (als Antwort auf: mlindqvist)

Re: Enforced displacement, tangential direction, more than 1 revolution

I'm somewhat not surprised. The quality of Simulate has seemed to take a hit ever since PTC shutdown the San Jose office (which housed the original creators of Mechanica that were still with the company) and shifted their responsibilities to India.

Aktionen ▾ — Gefällt mir (0)

Feedback to PTC

6. Outlook

- All models and error information shown in this presentation has been given to PTC R&D for examination and bug fixing
- Altran offers to report on the next year's SAXSIM about the progress PTC is doing with fixing all existing and new issues found
- We will provide a new revision "Finite Friction Contact 2.0" of the presentation on hand on the next year's SAXSIM if PTC provides by time a maintenance release of Creo 3.0 with fixed finite friction capabilities
- If requested by PTC, we will publish any documents provided to show the achievements in improving the engine immediately on the SAXSIM homepage for other user's information
- PTC is invited to support this process, preferably by hiring back the brightest sparks of the former RASNA Mechanica engine R&D team in California or by ramping up sufficiently skilled resources in India to bring the Creo Simulate engine back to an acceptable quality level and to be able to implement the necessary enhancements
- This means a magnitude of 10 experienced engine coders *with skills in structural mechanics and numerical solutions methods*, not 1!
- Please THINK BIG, don't try again just to fix the most critical issues with thinned out personnel in time spans of years! – With the former engine group in San Jose, bugs have been successfully fixed within one or at least two maintenance releases, and it was a pleasure to work with the code – now, it's not!

Part D: Appendix

Acknowledgement

Acknowledgement

Thanks to Richard B. King, PhD, for providing useful background information about the finite friction contact theory and software functionality

Note:

Richard King, co-founder of RASNA, first coder of Mechanica applied Structure and later principal coder at PTC Simulation R&D, now develops an own p-FEM code called "StressRefine", see www.stressrefine.com

StressRefine

Accurate Adaptive Stress Analysis

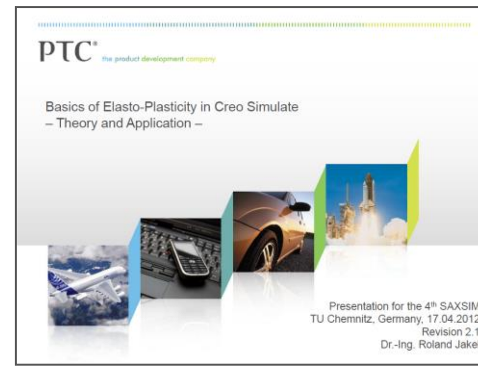
References (1)

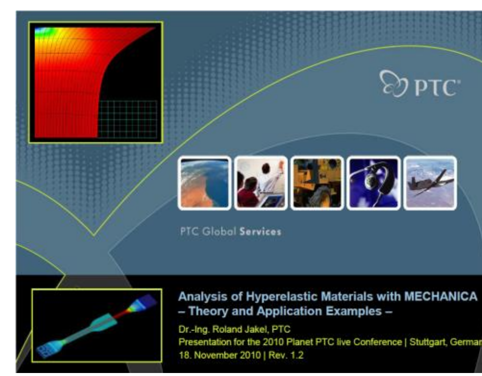
[1] R. Jakel: The New Contact with Friction Feature in Mechanica WF 4.0
–Theoretical Fundamentals and Application Examples –
English Version of the Presentation for the 1st SAXSIM, TU Chemnitz, 28-April-2009;
for the German version, see www.saxsim.de,
for the English version, see
https://www.ptcusercommunity.com/message/424876#424876)

[2] R. Jakel: Basics of Elasto-Plasticity in Creo Simulate
– Theory and Application –
Presentation for the 4th SAXSIM, TU Chemnitz, Germany, 17.04.2012, Revision 2.1

[3] R. Jakel: Analysis of Hyperelastic Materials with Mechanica – Theory and Application Examples –
Presentation for the 2010 Planet PTC live Conference, Stuttgart, Germany, 18. November 2010, Rev. 1.2

[4] Crisfield, M: Nonlinear Finite Element Analysis of Solids and Structures; Wiley, 1991, page 254 ff





altran

References (2)

[5] Abaqus 6.12 Theory Manual, 5.1 Contact modeling, 5.1.2 Finite sliding interaction between deformable bodies
http://xn--90ajn.xn--p1ai:2080/v6.12/books/stm/default.htm

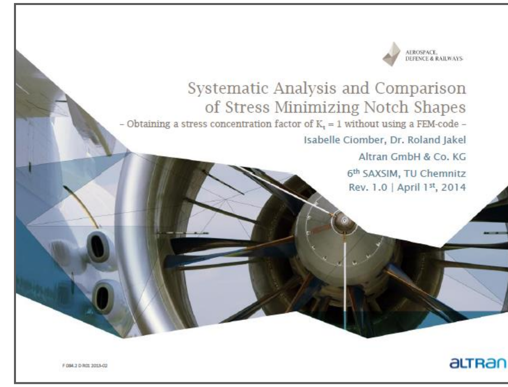
[6] I. Ciomber, R. Jakel: Systematic Analysis and Comparison of Stress Minimizing Notch Shapes; Presentation for the 6th SAXSIM, TU Chemnitz, Germany, 01-April-2014, Revision 1.0

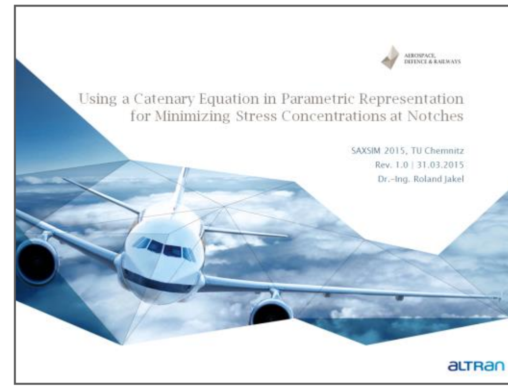
[7] R. Jakel: Using a Catenary Equation in Parametric Representation for Minimizing Stress Concentrations at Notches;
Presentation for the 7th SAXSIM, TU Chemnitz, Germany, 31-March-2015, Revision 1.0

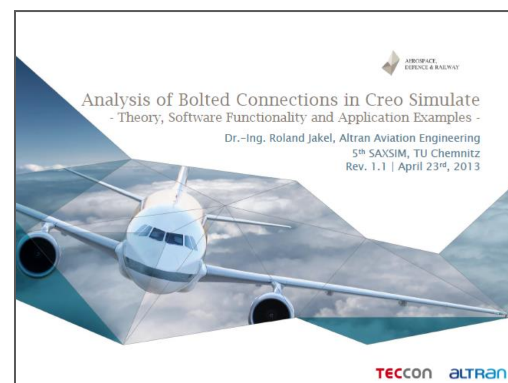
[8] R. Jakel: Analysis of Bolted Connections in Creo Simulate: Theory, Software Functionality and Application Examples

Presentation for the 5th SAXSIM, TU Chemnitz, Germany, 23-April-2013, Revision 1.1

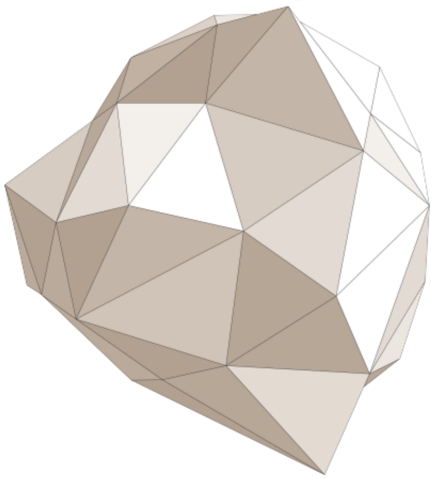
All these presentations can be found in the archive of www.saxsim.de







INNOVATION MAKERS



Dr.-Ing. Roland Jakel
Senior Consultant, Structural Simulation

Phone: +49 (0) 421 / 557 18 4111
Mobile: +49 (0) 173 / 889 04 19
roland.jakel@altran.com

altran